Surface Editing Tools and Features

Our Designer Elements program provides several editing tools in the **Surface Editing** tool palettes. The tools allow you to perform such operations as intersecting existing surfaces and solids with other entities, projecting curves onto surfaces, adding to or subtracting from and intersecting 2D planar surfaces as well as joining or trimming surfaces and matching surfaces. Our Designer Elements program also includes features for reshaping surfaces through control vertex modification.

The following topics are covered in this chapter:

- **Solid Editing** tools
- Surface Control Vertex Modification
- Surface Evaluation

Solid Editing Tools

Many of the editing tools require a two step process for their usage. The first step after choosing the tool may be to select one or more objects. The second step performs the editing operation. Once you the complete the first step you cannot select more objects to include in the same operation. To select additional objects reset the tool and start again.

Note: These tools do not support the use of grouped objects.

The geometric characteristics listed in the Edit Objects dialog box after editing an object using these tools are the same as the object's standard characteristics. For example, if a curve/surface intersection is performed, the Edit Objects dialog would display characteristics of the resulting curve.

Selecting Objects within the Editing Tools

With these editing tools you are asked to select the objects for performing an operation. When you select the tool and move the cursor into the drawing area it becomes a hollow selection arrow (shown here).

When you finish selecting the pointer becomes a general tool pointer (shown here.)

This hollow selection arrow must not be confused with the **Selection** tool.

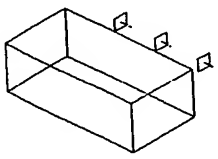
Plane/Surface Intersection Tool

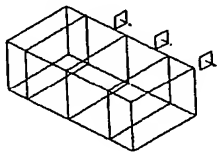
The **Plane/Surface Intersection** tool calculates the intersection of an infinite plane with a curve, surface or solid. The resulting cut is automatically associative with the plane and the object so that if you modify either entity, the cut will update (**Cobalt™ and Xenon™ only**).

Auto Layers Option

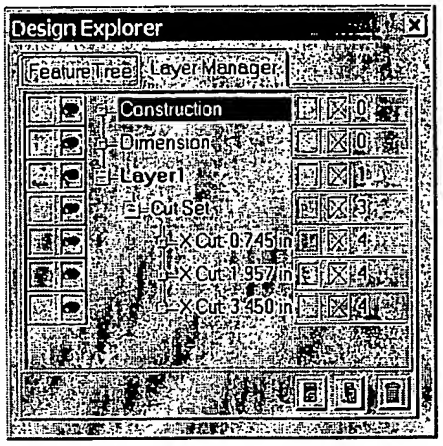
Tip:
When the *Auto Layers* option is used with the *Increment Layer* command (ALT+0 - Windows, z +0 - Macintosh) and Decrement Layer (ALT+9-Windows, z +9-Macintosh), to change layers, you have a convenient way of examining every 2D cut of a 3D model.

This tool provides an *Auto Layer* option in the Message Line. When this option is enabled, our Designer Elements program puts each section cut on a separate layer and names the layer according to the location of the cut. For example, the graphics here show a solid block with three intersecting planes.





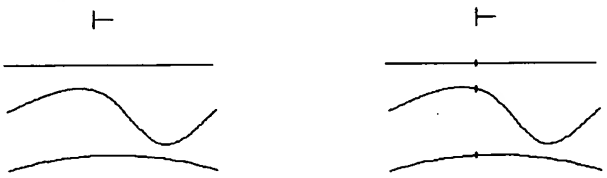
With the *Auto Layers* option enabled each section is placed on a separate layer. The Layer Manager graphic below shows the new layers.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when using this tool.

Intersecting a Plane with a Curve

When you intersect a plane with a curve a point intersection results. The left graphic here shows a plane and three curves. The right graphic displays the intersection points for the plane and the curves.



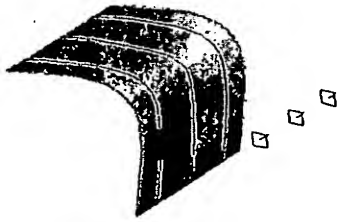
1. Select the tool in the tool palette. The Message Line reads: *Plane/Surface Intersection: Pick the planes to slice objects with [Shift+Extend].*
2. Click the plane you want to use to intersect the curve. Hold down the SHIFT key to pick more than one plane. The Message Line reads: *Plane/Surface Intersection: Pick objects to intersect with plane [Shift=Extend].*
3. Click the curves that you want to intersect with the plane.

   The intersection point displays. The point is associative to the plane and curve. Modifying either the plane or the curve automatically updates the location of the point.

There are no entries in the Status Line.

Intersecting a Plane with a Surface

When you intersect a plane with a surface a curve intersection results. If the curve is a spline, it is a vector spline capable of being edited.



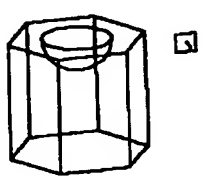
1. Select the tool in the tool palette. The Message Line reads: *Plane/Surface Intersection: Pick planes to slice objects with [Shift=Extend].*
2. Click the plane you want to use to intersect the surface. Hold down the SHIFT key to pick more than one plane. The Message Line reads: *Plane/Surface Intersection: Pick objects to intersect with plane [Shift=Extend].*
3. Click the surface that you want to intersect with the plane.

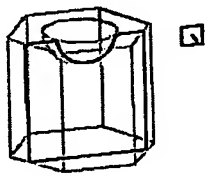
   The intersection curve displays. The curve is associative to the plane and surface. Modifying either the plane or the surface automatically updates the location of the curve (**Cobalt™ and Xenon™ only**).

There are no entries in the Status Line.

Intersecting a Plane with a Solid

When you intersect a plane with a solid a curve intersection results. If the curve is a spline, it is a vector spline capable of being edited.





1. Click the tool in the tool palette. The Message Line reads: *Plane/Surface Intersection: Pick planes to slice objects with [Shift=Extend].*

2. Click the plane you want to use to intersect the solid. Hold down the SHIFT key to pick more than one plane. The Message Line reads: *Plane/Surface Intersection: Pick objects to intersect with plane [Shift=Extend].*
3. Click the solid that you want to intersect with the plane.

   The intersection curve displays. The curve is associative to the plane and solid. Modifying either the plane or the solid automatically updates the location of the curve **(Cobalt™ and Xenon™ only)**.

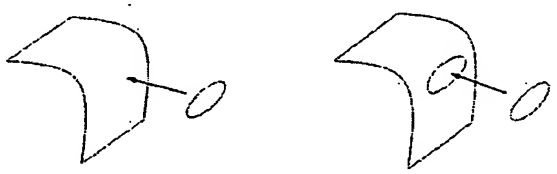
There are no entries in the Status Line.

Curve/Surface Projection Tool

The **Curve/Surface Projection** tool projects a curve along a direction vector onto a surface or a solid. You specify the direction vector by clicking two points. The new curve is associative to the object. When you modify the object the projected curve will automatically update **(Cobalt™ and Xenon™ only)**.

Projecting a Curve to a Surface

With this tool you can project a curve onto a surface. If the curve is a spline, it is a vector spline capable of being edited.



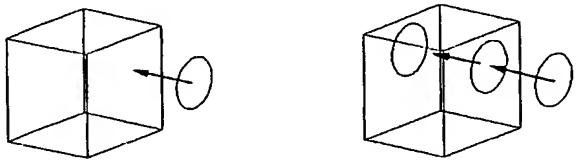
1. Select the tool in the tool palette. The Message Line reads: *Curve/Surface Projection: Pick curves to project onto surface [Shift=Extend].*
2. Click the curve to project onto the surface (it does not highlight). Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Projection: Pick ONE surface or solid for curve projections [Shift=Extend].*
3. Click the surface or solid onto which you want to project the curve.
4. Enter 2 points for the projection vector as directed by the Message Line.

   The curve projects onto the surface.

There are no entries in the Status Line.

Projecting a Curve to a Solid

With this tool you can project a curve onto a solid. If the curve is a spline, it is a vector spline capable of being edited.



1. Click the tool in the tool palette. The Message Line reads: *Curve/Surface Projection: Pick curves to project onto surface [Shift=Extend].*
2. Click the curve to project onto the solid (it does not highlight). Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Projection: Pick ONE surface or solid for curve projections [Shift=Extend].*
3. Click the solid onto which you want to project the curve.
4. Enter 2 points for the projection vector as directed by the Message Line.

   The curve projects onto the solid.

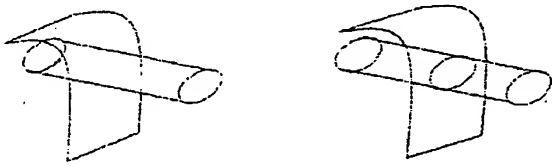
There are no entries in the Status Line.

Surface/Surface Intersection Tool

The **Surface/Surface Intersection** tool calculates the intersection of two existing surfaces or a surface and a solid. Select the two objects and the tool creates one or more curves. The curves are associative with both objects (**Cobalt™ and Xenon™ only**). When you change either object the intersection curve will update. To remove the associative link, use ***Edit>Remove Links***.

Intersecting a Surface with a Surface

Use this tool to intersect two surfaces producing a curve intersection. If the curve is a spline, it is a vector spline capable of being edited.

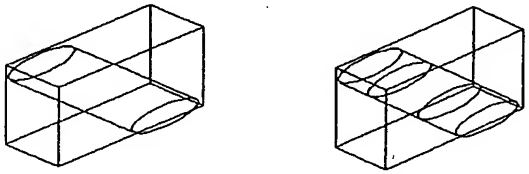


1. Select the tool in the tool palette. The Message Line reads: *Surface/Surface Intersection: Pick two surfaces or solids for intersection.*
2. Click on the two surfaces. An intersection curve displays.

There are no entries in the Status Line.

Intersecting a Surface with a Solid

Use this tool to intersect a surface with a solid.



1. Click the tool in the tool palette. The Message Line reads: *Surface/Surface Intersection: Pick two surfaces or solids for intersection [Shift=Extend].*
2. Click the two surfaces. An intersection curve displays.

There are no entries in the Status Line.

Curve/Surface Intersection Tool

The **Curve/Surface Intersection** tool calculates the intersection of a curve and a surface or solid. The tool creates a collection of point objects. The points are associative to both the curve and object **(Cobalt™ and Xenon™ only)**.

Intersecting a Curve with a Surface

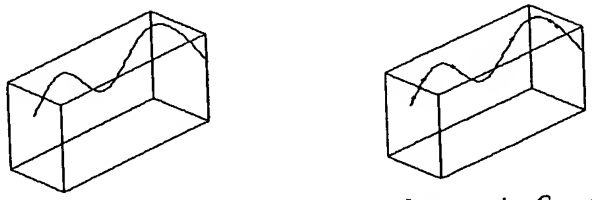
You can intersect a curve with a surface with this tool.

1. Select the tool in the tool palette. The Message Line reads: *Curve/Surface Intersection: Select curves to intersect surface/solids [Shift=Extend].*
2. Click the curve to intersect the surface. Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Intersection: Select surfaces or solids to intersect with curve [Shift=Extend].*
3. Click the surface to intersect with the curve. The intersection points display.

There are no entries in the Status Line.

Intersecting a Curve with a Solid

You can intersect a curve with a solid with this tool.



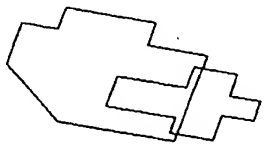
1. Select the tool in the tool palette. The Message Line reads: *Curve/Surface Intersection: Select curves to intersect surface/solids [Shift=Extend].*
2. Click the curve to intersect the solid. Hold down the SHIFT key to pick more than one curve. The Message Line reads: *Curve/Surface Intersection: Select surfaces or solids to intersect with curve [Shift=Extend].*
3. Click the solid to intersect with the curve. The intersection points display.

There are no entries in the Status Line.

Boolean Surface Tools

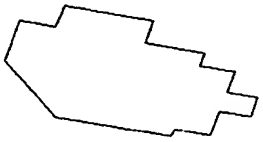
Our Designer Elements program supports 2D boolean operations on planar surfaces created with the cover surface command. With the **Boolean Surface** tool you can add surfaces together, subtract surfaces from one another, and find the intersection of two or more surfaces. You can use the new curves created from boolean operations for 2D property analysis or as profiles for solid extrusions, sweeps and lathe operations.

You have three tools for performing boolean operations: **Add Boolean Surface**, **Subtract Boolean Surface** and **Intersect Boolean Surface**.



For the purposes of explaining these tools the objects to the right are used.

Add Boolean Surface Tool



With this tool you add two objects to create a third object.

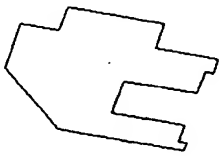
1. Select the **Boolean Surface** tool.
2. Select the **Add Boolean Surface** tool in the Message Line. The Message Line reads: *Boolean Surface: Pick planar surface to add to [Shift=Extend].*
3. Select the surface to which you will add another. The Message Line reads: *Boolean Surface: Pick planar surface to add [Shift=Extend].*
4. Select the surface to be added.

There are no entries in the Status Line.

Tip:

If you want to get volume, density and other properties of geometry in your drawing, you can combine them with this tool whether or not they are connected.

Subtract Boolean Surface Tool



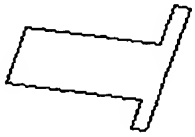
With this tool you subtract one surface from another.

1. Select the **Boolean Surface** tool.

2. Select the **Subtract Boolean Surface** tool in the Message Line. The Message Line reads: *Boolean Surface: Pick planar surface to subtract from [Shift=Extend].*
3. Select the surface from which you will subtract another. The Message Line reads: *Boolean Surface: Pick planar surface to subtract [Shift=Extend].*
4. Select the surface to be subtracted. One surface is subtracted from the other.

There are no entries in the Status Line.

Intersect Boolean Surfaces Tool



With this tool you retain the intersecting portion of two surfaces.

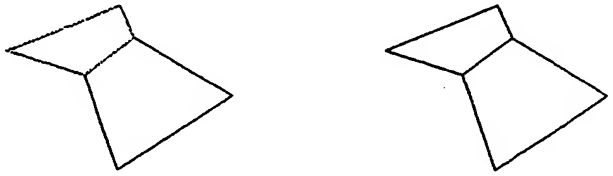
1. Select the **Boolean Surface** tool.
2. Select the **Intersect Boolean Surface** tool in the Message Line. The Message Line reads: *Boolean Surface: Pick planar surface to intersect with [Shift=Extend].*
3. Select the surface with which you will intersect another. The Message Line reads: *Boolean Surface: Pick planar surface to intersect [Shift=Extend].*
4. Select the surface to be intersected. The intersecting portion of the surfaces is displayed.

There are no entries in the Status Line.

Connect Surface Tool

The **Connect Surface** tool merges two surfaces into one. The resulting surface retains the original surfaces mathematical definitions, but a selection operation treats both surfaces as one. Unlike the **Add Boolean Surface** tool the **Connect Surface** tool can be used with non-planar surfaces. The left graphic below displays two

separate but connecting surfaces. The graphic on the right displays one surface that retains the characteristics of the individual surfaces.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when using this tool.

Using the Connect Surface Tool

1. Click the **Connect Surface** tool in the tool palette. The Message Line reads: *Connect Surface: Pick two surfaces to connect [Shift=Extend].*
2. Click the two surfaces that you want to connect. The surfaces become one surface but retain their individual mathematical definitions.

There are no entries in the Status Line.

Split Surface Tool

Use the **Split Surface** tool to cut away portions of a surface. You can trim a surface to another surface, curve or solid. Unlike the **Subtract Boolean Surface** tool the **Split Surface** tool can be used with non-planar surfaces.

Using the Tool to Split a Surface with a Surface

You can use this tool to trim a surface to a surface.

1. Select the **Split Surface** tool in the tool palette. The Message Line reads: *Split Surface: Pick surface to split [Shift=Extend].*
2. Select the surface that you want to split. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with [Shift=Extend].*
3. Select the entity that defines the trim area. Our Designer Elements program splits the surface at the intersection of the two entities. Your objects won't appear differently on your screen. Delete any portions that you don't need.

There are no entries in the Status Line.

Using the Tool to Split *the Surface to a Curve*

You can use this tool to trim a surface to a curve.

Tech Note:

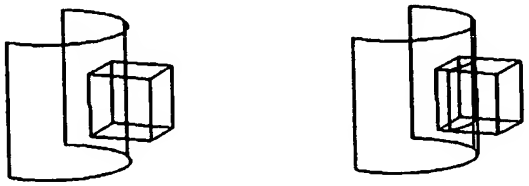
The Split Surface Tool will not work with grouped surfaces.

1. Select the **Split Surface** tool in the tool palette. The Message Line reads: *Split Surface: Pick surface to split [Shift=Extend].*
2. Select the surface that you want to trim. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with [Shift=Extend].*
3. Select the entity that defines the trim area. Our Designer Elements program splits the surface at the intersection of the surface and curve. Your objects won't appear differently on your screen. Delete any portions that you don't need.

There are no entries in the Status Line.

Using the Tool to Split *a Surface with a Solid*

You can use this tool to split a surface to a solid.



1. Select the **Split Surface** tool in the tool palette. The Message Line reads: *Split Surface: Pick surface to split [Shift=Extend].*
2. Select the surface that you want to split. The Message Line reads: *Split Surface: Pick curves, surface, or solid to split with [Shift=Extend].*
3. Select the entity that defines the split area. Our Designer Elements program splits the surface at the intersection of the surface and the solid. Your objects won't appear differently on your screen. Delete any portions that you don't need.

There are no entries in the Status Line.

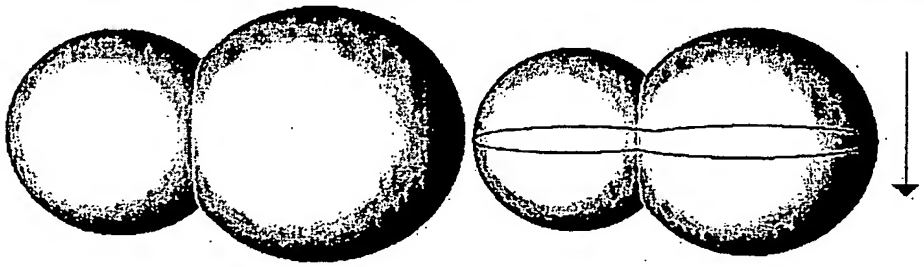
Silhouette Curves Tool

The **Silhouette Curves** tool creates curves from the bounding edge of an object relative to a user-defined projection direction. The curves produced by this tool are, in most cases, non-planar. This tool is usefule for próducing injection mold drawings.

Using the Silhouette Curves Tool

1. Select the Silhouette Curves tool, The Message Line reads: *Silhouette Curves: Pick surface/solids for silhouettes. [Shift = Extend]*
2. Select the objects on which you want silhouette curves. The message line now reads, *Silhouette Curves: Specify two points for view direction.*

The example below shows the progression of use with the Silhouette Curves tool.

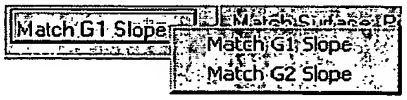


Local Surface Tools

Match Surface Tool

This tool gives you the ability to impose a smooth transition from one surface to another neighboring surface. It also allows you perform greater surface manipulation by inserting a knot and elevating the degree of the surface.

There are five options available for this tool: *Match G1 Slope*, *Match G2 Slope*, *Insert Knot*, *Elevate Degree* and *Untrim Surface*. These tools are accessed through the pull-down menu in the Message Line.



Rules and Notations for using the Match Surface Tool and Options

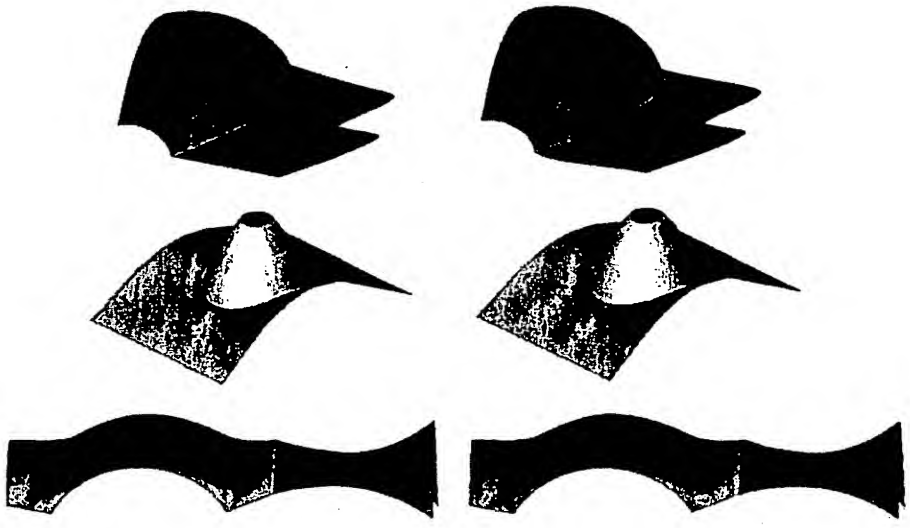
- Surfaces can not be trimmed surfaces. (Any surface that has had a relimit or another editing operation conducted on it is a trimmed surface.) If you want to match untrimmed surfaces, you must first untrim them using the *Untrim Surface* option of this tool before matching them. See a later section about this option.
- Surfaces must share the same edge being within 0.001 inches or 0.0254 mm of each other.
- The direction and slope of the surfaces do not matter.
- Matching two surfaces with the *G1 Slope* or *G2 Slope* options adjusts the slope of the first surface to that of the second. To reduce the influence of the matched slope on the surface, elevate its degree using the *Elevate Degree* option of this tool before matching the surfaces. See a later section about this option.

Match G1 Slope Option

Use this option to match two untrimmed surfaces with G1 continuity. G1 matched surfaces are tangent continuous at their shared edge. The graphics on the left below

are the two original surfaces. The graphics on the right below are the same surfaces using the *G1 Slope* option.

Tech Note:
G1 stands for geometric continuity matched to the first derivative, tangency. G1 matched surfaces are tangent continuous.



Using the G1 Slope Surface Option

1. Select the **Match Surface** tool.

2. From the pull-down menu in the Message Line, select *Match G1 Slope*. The Message Line reads: *Match Surface: Pick surface edge to modify tangency.*
3. Click on the edge of the surface you want to modify.

   Be sure to click on the edge and not somewhere on the body of the surface. The entire surface highlights. The Message Line now reads, *Match Surface: Pick surface edge to match.*
4. Click on the surface edge to which you are matching the first surface.

   The slope of the first surface edge now matches the second surface edge with a G1 continuity. You can verify this continuity by using the surface evaluation feature. In a surface evaluation plot, stripes represent the slope of the surface.

These stripes line up from the first surface to the second surface for G1 continuous surfaces.

See the Surface Evaluation section at the end of this chapter for information.

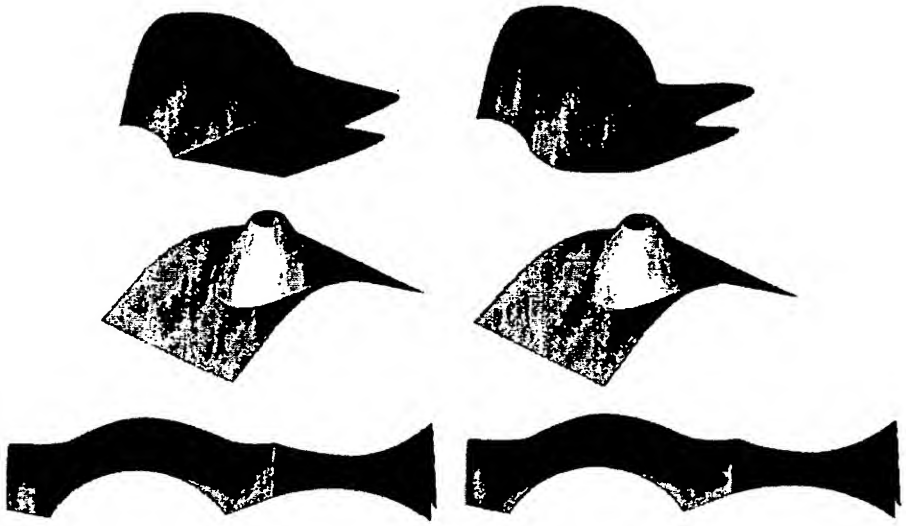
There are no entries in the Status Line.

Tech Note:

G2 stands for geometric continuity matched to the first and second derivative tangency and curvature continuous. G1 matched surfaces are tangent and curvature continuous.

Match G2 Slope Option

Use this option to match two untrimmed surfaces with G2 continuity. G2 matched surfaces are tangent and curvature continuous at their shared edge. The graphics on the left below are the two original surfaces. The graphics on the right below are the same surfaces using the *G2 Slope* option.



Using the G2 Slope Surface Option

1. Select the **Match Surface** tool.
2. From the pull-down menu in the Message Line, select *Match G2 Slope*. The Message Line reads: *Match Surface: Pick surface edge to modify tangency.*
3. Click on the edge of the surface you want to modify.

   Be sure to click on the edge and not somewhere on the body of the surface. The entire surface highlights. The Message Line now reads, *Match Surface: Pick*

surface edge to match.

4. Click on the surface edge to which you are matching the first surface.

The slope of the first surface edge now matches the second surface edge with a G2 continuity. You can verify this continuity by using the surface evaluation feature. In a surface evaluation plot, stripes represent the slope of the surface. These stripes line up from the first surface to the second surface for G2 continuous surfaces.

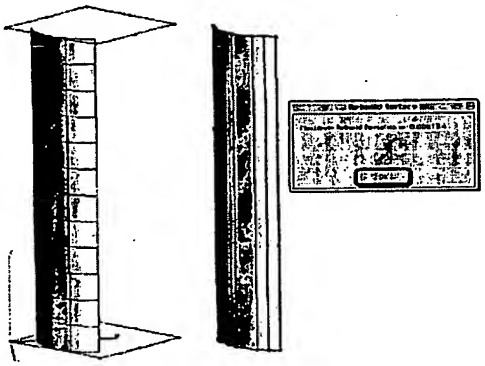
See the Surface Evaluation section at the end of this chapter for information.

There are no entries in the Status Line.

Rebuild Surface Tool

The Rebuild Surface tool reconstructs an approximating surface to the surface to rebuild. The tightest tolerance achieved is displayed in a dialog box. If the tolerance achieved is not tight enough use undo to put the surface back in its orginal condition.

The Rebuild Surface tool is useful for converting analytics to cubic NURBs, trimmed surfaces to untrimmed surfaces, and repairing surfaces. The rebuild tool is limited to surfaces with 3 or 4 sides.



Using the Rebuild Surface Tool

1. Select the Rebuild surface tool. The Message Line reads: Rebulid Surface:Pick surface to rebuild.
2. Click the surface to rebuild.

There are no status line entries for this command.

Untrim Surface Tool

Using this option removes all trim boundaries on a surface face type surface. The left graphic below shows a trimmed surface. The right graphic shows the surface after this option is used.

Using the Untrim Surface Option

1. Select the **Untrim Surface** tool.

2. From the pull-down menu in the Message Line, select *Untrim Surface*. The Message Line reads: *Untrim Surface: Pick surface to untrim.*

   The surface untrims.

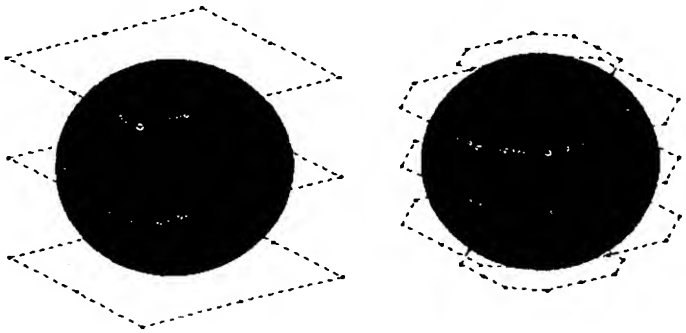
There are no entries in the Status Line.

Elevate Surface Tool

This option elevates the degree of a surface introducing more control points. These control points can be used to modify the surface.

NURB surfaces are defined by polynomial equations. The most basic equation possible is used to define the surface. Where cylindrical and spherical shapes are defined by second degree polynomials, skin, cover and net surfaces are defined by third degree polynomials due to their complexity. Shapes defined by higher degree polynomicals have more control points than those defined by lesser degree polyno-

mials. This option raises the degree level for a shape providing more control points for surface manipulation.



Surfaces can be elevated up to the 22nd degree. However, it is recommended that you never elevate a curve higher than 9 degrees. Degree elevation is useful when trying to match surfaces and minimize the effect of the match on the surface.

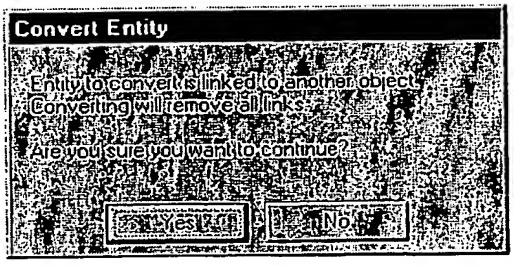
Using the Elevate Surface Tool

1. Select the surface to which you want to add control points.
2. Choose ***Window>Edit Objects*** to verify that the surface is a surface face.

   If your surface is a surface face continue with step 5.
3. Choose ***Edit>Change Object Type***.
4. Select the Surface option and click OK.

   If the surface you selected is linked to another object, the following warning box appears.

   Click Yes to continue conversion.



5. Choose ***Edit>Show Points*** to display the surface control points.
6. Select the **Elevate Surface** tool.

7. From the pull-down menu in the Message Line select: *Elevate Degree*. The Message Line reads: *Elevate Surface: Pick surface to elevate.*
8. Select the surface.

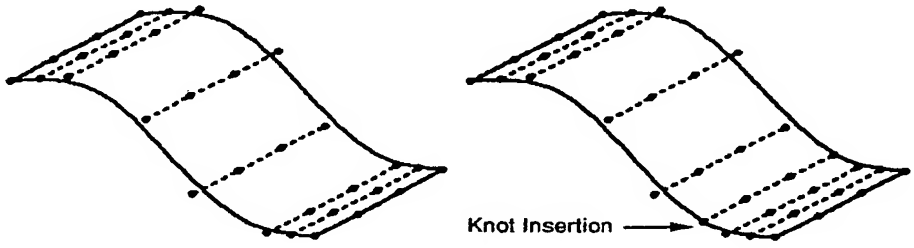
   The surface elevates to the next degree.

There are no entries in the Status Line.

Insert Knot Tool

This tool allows you to add a row or column of control points to a surface. The surface must be untrimmed and the *surface face* type. Net, skin and cover surfaces must be converted into the surface face type using the *Change Object Type* command in the Edit menu. You can verify the surface type by double-clicking on the surface to display the Edit Objects dialog box.

Knot insertion is valuable when you want to adjust a surface shape at a location where there are no control points. Knot insertion is also useful when trying to match surfaces and minimize the effect of the match on the surface. The left graphic below displays the control points of the original surface. The right graphic shows an additional row of control points at the right end of the surface.



After inserting control points using this option, you may notice that other control points may have shifted. This is automatically done to preserve the surface shape.

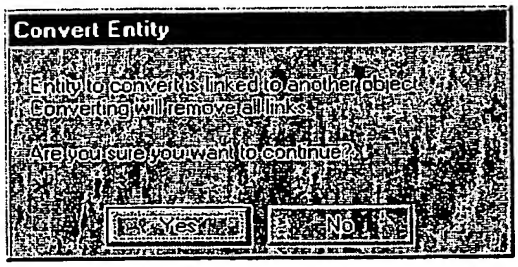
Using the Insert Knot Tool

1. Select the surface to which you want to add control points.
2. Choose ***Window>Edit Objects*** to verify that the surface is a surface face.

   If it is continue on with step 5.

3. Choose ***Edit>Change Object Type.***
4. Select the Surface option and click OK.

   If the surface you selected is linked to another object, the following warning box appears.

   Click Yes to continue conversion.
5. Choose ***Edit>Show Points*** to display the surface control points.
6. Select the Insert Knot tool.

   
7. From the pull-down menu in the Message Line select *Insert Knot.* The Message Line reads: *Insert Knot: Pick surface to insert a new knot.*
8. Select the surface. The Message Line now reads: *Insert Knot: Specify location for new knot.*
9. Select a location along an edge of the surface for the new row or column of knots.

   A new series of knots are added. If you want to add another row or column of knots you must select the surface again.

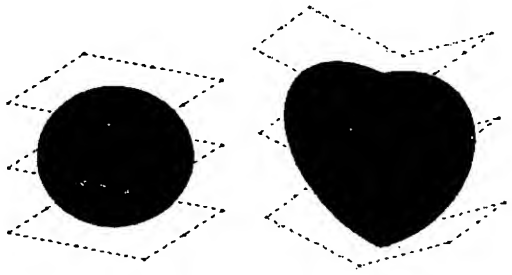
There are no entries in the Status Line.

Surface Control Vertex Modification

Our Designer Elements program supports the ability to manipulate the control points for all surface face type NURB surfaces. All other surfaces like net, skin and cover, must be converted into the surface face type using the *Change Object Type* command. Remember that using the *Change Object Type* command removes all links from the surface.

After showing points on the selected surface you can select any point and move it to a new location. With this ability you can create such things as creases and other surface variations. If surface smoothness is a concern, be sure to use the surface evaluation plot available through the Verify menu or the advanced rendering features. See the next section.

The left graphic below shows a sphere converted into a surface and displaying its control points. The right graphic shows the surface after control point modification creating a heart shape.

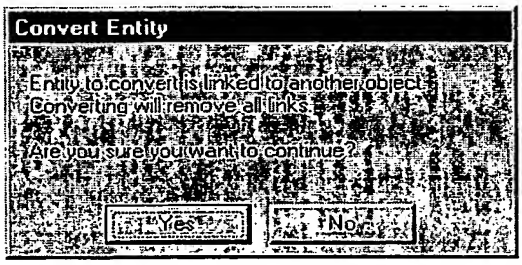


Performing a Control Vertex Modification

1. Select the surface you want to modify.
2. Choose ***Window>Edit Objects*** to verify that the surface is a surface face.

   If it is continue on with step 5.
3. Choose ***Edit>Change Object Type***.
4. Select the *Surface* option and click OK.

   If the surface you selected is linked to another object the following warning box appears:

   Click Yes to continue conversion.
5. Choose ***Edit>Show Points*** to display the surface control points.
6. Using the **Selection** tool, drag the control points as desired.

Surface Evaluation

Our Designer Elements program gives you the ability to evaluate the smoothness of your surfaces. This is useful for identifying surface irregularities. The curvature evaluation is accomplished by placing your surface inside a brightly lit imaginary cylin-

der with longitudinal stripes. The cylinder stripes are reflected onto the surface to convey the surface smoothness. This can especially important when you match surfaces using this **Match Surface** tool. The graphics here show the surface evaluation for two matched surfaces using no surface matching (G0), G1 and G2 surface matching.

G0 Matched Surfaces

G1 Matched Surfaces

G2 Matched Surfaces

Notice how the bands of black differ from graphic to graphic. In the G0 Matched Surfaces graphic (no surface match), the right surface bands are parallel. In the G1 Matched Surfaces graphic, the right surface bands are not as parallel showing that the surface was adjusted to match the left surface. In the G2 Matched Surface graphic, the right surface bands show how the surface has even further adjusted to match the left surface.

You can evaluate your surfaces either through the Verify menu using the *Surface Analysis* commands or through the advanced rendering feature.

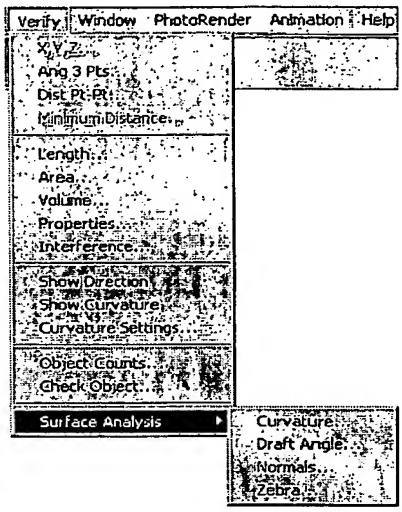
Evaluation Through the Verify Menu

Refferal Note:

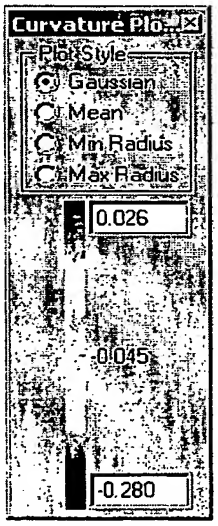
This section is duplicated in the Verify Menu section of Chapter 24.

The *Surface Analysis* commands allow you to quickly evaluate the smoothness of one or more surfaces. All analysis colors are preset with these commands. If you would like to define your own settings use the surface evaluation shader through the Advanced Rendering feature. See "Evaluation Through the Advanced Rendering Feature" on page 34.

You have four surface analysis commands, *Curvature, Draft Angle, Normals* and *Zebra*. When you choose ***Verify>Surface Analysis***, a submenu appears containing the commands.



Curvature Command



This command creates a curvature plot of your surface. When you select the command the Curvature Plot dialog box appears containing: Analysis Styles, Histogram, Color Spectrum and Histogram Data Fields.

The dialog box contains the following elements:

Plot Style — This section contains the styles for the analysis and include: *Gaussian, Mean, Min Radius* and *Max Radius*. A description and illustration of each style is provided in the next section.

Histogram — Located to the left of the color spectrum in the dialog box these horizontal bars represent the frequency of a curvature smoothness (change in a curve over the change in curvature) using the color spectrum. The length of the bar

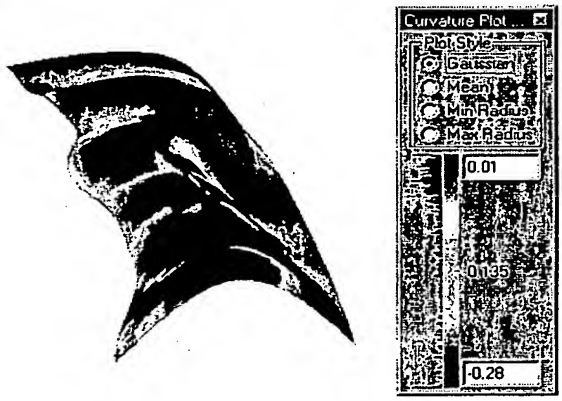
represents the frequency. Our Designer Elements program calculates this histogram and displays it so that the entire graph fits into the dialog box area.

Color spectrum bar

The section displays the color spectrum used to create the histogram.

Histogram data fields

The data fields display the maximum and minimum values used to calculate the histogram. When you initially select the command, our Designer Elements program scans the surface(s) and sets the min and max fields for curvature. If you enter different values in the fields such that a large number of curvature values fall outside of the specified range, a red line appears at the end(s) where the values fall. All values are still calculated even though they are not displayed due to the specified range. In the graphic here the large number of values fall outside the maximum value of 0.006.



If you make changes to the values our Designer Elements program waits for two seconds of non-action before recalculating the histogram, giving you time to change the values before the image is rendered again.

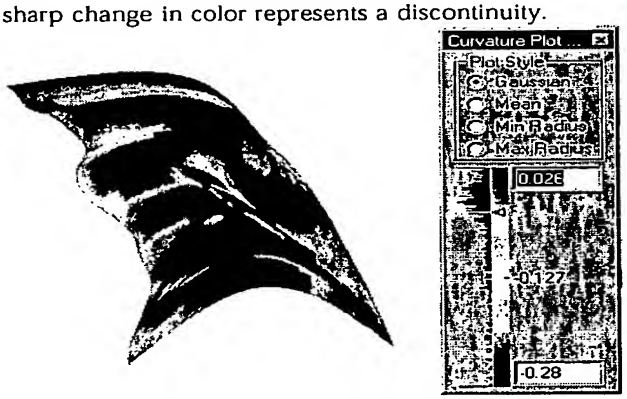
You can return to the range initially displayed by clicking on the selected option again. (The option does not deselect when you do this.) Windows users can also reset the maximum and minimum values by right mouse clicking near the respective end of the spectrum.

Plot Styles

You have four curvature plot styles to choose from when you select the *Curvature* command: *Gaussian*, *Mean*, *Min Radius* and *Max Radius*.

Gaussian

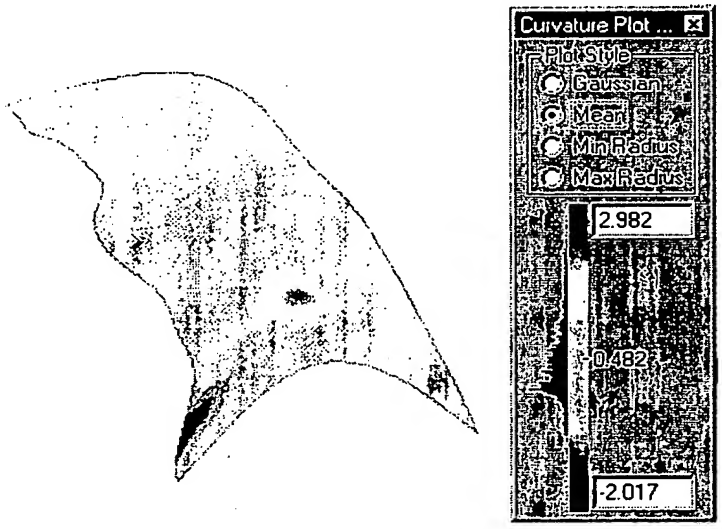
Selecting this option creates a Gaussian curvature plot on your selected surface. The plot registers the change in a curve over the change in curvature. Mathematically, the Gaussian value is the product of the kmin (minimum radius curvature) and kmax (maximum radius curvature) of each vertex. (The letter "k" refers to the curvature.) Any sharp change in color represents a discontinuity.



Mean

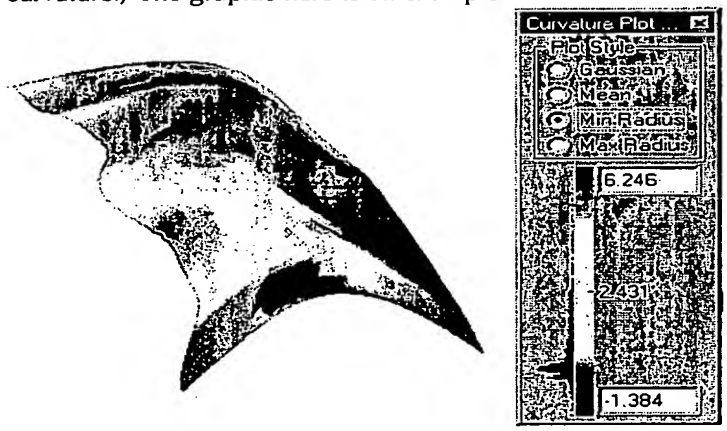
Selecting this option creates a mean curvature plot. Mathematically, the mean value is the average of the kmin (minimum radius curvature) and kmax (maximum radius

curvature) of each vertex. (The letter "k" refers to the curvature.) The graphic here is an example.



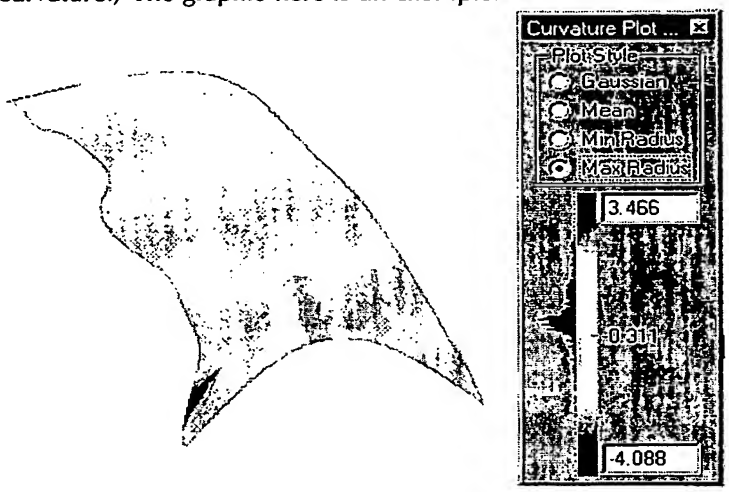
Min Radius

Selecting this option creates minimum radius curvature plot. Mathematically, the plot is the kmin values (minimum radius curvature) of each vertex. (The letter "k" refers to the curvature.) The graphic here is an example.



Max Radius

Selecting this option creates maximum radius curvature plot. Mathematically, the plot is the kmax value (maximum radius curvature) of each vertex. (The letter "k" refers to the curvature.) The graphic here is an example.



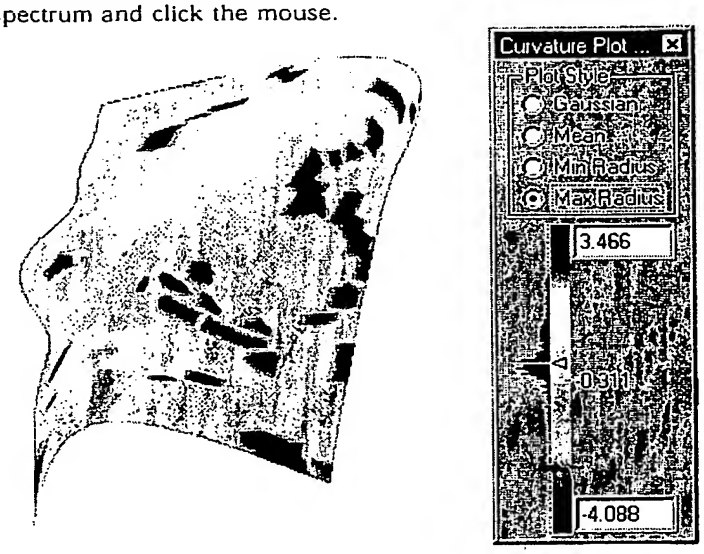
Using a Curvature Surface Analysis Command

1. Select the surface.
2. Choose ***Verify>Surface Analysis>Curvature***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.
3. Select a Plot style, if the desired style is not already selected.

   A new curvature analysis appears.

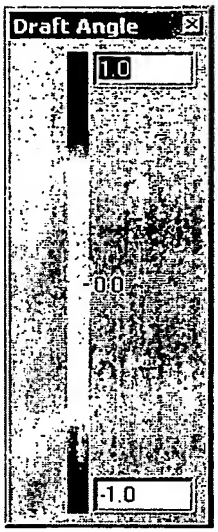
4. To display a certain analysis/color area, place the pointer at the location over the color spectrum and click the mouse.



Notice that a triangular indicator appears at the selected location and the related color highlights in the histogram. The same color highlights in black on your surface.

5. To remove the triangular indicator click in the dialog box outside of the color spectrum bar.
6. Change the histogram values in the data fields and the histogram and analysis automatically recalculates.

Draft Angle



This command evaluates the drafts of an object for molding purposes. When you select this command the Draft Angle dialog box appears with the draft angle analysis containing the histogram with the analysis, the color spectrum and histogram data fields.

The dialog box contains the following elements:

Histogram

Located to the left of the color spectrum in the dialog box these horizontal bars represent the frequency of a draft angle using the color spectrum. The length of the bar represents the frequency.

Our Designer Elements program calculates this histogram and displays it so that the entire graph fits into the dialog box area.

Color spectrum bar

The section displays the color spectrum used to create the histogram.

Histogram data fields

The data fields display the maximum and minimum values used to calculate the histogram. When you initially select the command our Designer Elements program scans the surface(s) and sets the min and max fields for the draft angle analysis.

If you enter different values in the fields such that a large number of values fall outside of the specified range a red line appears at the end(s) where the values fall. All values are still calculated even though they are not displayed due to the specified range.

If you make changes to the values our Designer Elements program waits for two seconds of non-action before recalculating the histogram, giving

you time to change the values before the image is rendered again.

You can return to the range initially displayed by clicking on the selected option again. (The option does not deselect when you do this.) Windows users can also reset the maximum and minimum values by right mouse clicking near the respective end of the spectrum.

The graphic here is an example of a draft angle analysis.

Using the Draft Angle Command

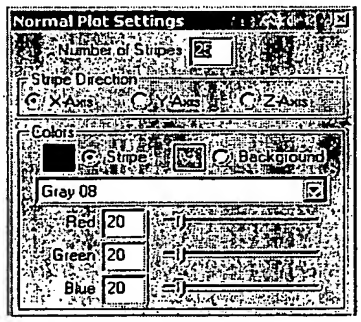
1. Select the object.
2. Choose ***Verify>Surface Analysis>Draft Angle***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.
3. To display a certain analysis/color area place the pointer at the location over the color spectrum and click the mouse.

   A triangular indicator appears at the selected location and the related color highlights in the histogram. The same color highlights in black on your surface.
4. To remove the triangular indicator click in the dialog box outside of the color spectrum bar.
5. Change the histogram values in the data fields and the histogram and analysis automatically recalculates.

Normals

This command creates a normals plot of your surface. When you select the command the Normal Plot Settings dialog box appears.



The dialog box contains the following elements:

Number of Stripes — This data field contains the number stripes that appear on your surface.

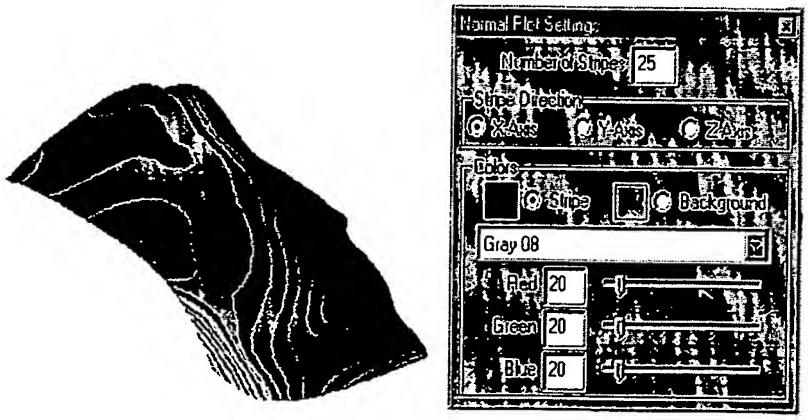
Stripe Direction — This section provides three direction options: X-Axis, Y-Axis and Z-Axis.

Colors — This sections contains the *Stripe Color* option and swatch, *Background Color* option and swatch; the color pull-down menu and RGB data fields and sliders.

Stripe Color option: When selected you can set the stripe color from the color pull-down menu or the RGB fields.

Background Color option: When selected you can set the surface background color from the color pull-down menu or the RGB fields.

This graphic shows an example of a normal surface analysis.



Using the Normal Command

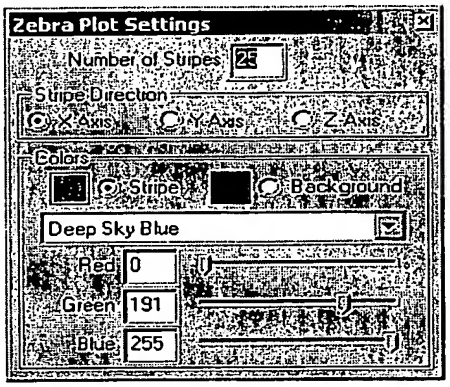
1. Select the surface.
2. Choose ***Verify>Surface Analysis>Normals***.

   The dialog box displays with the analysis. Your geometry also displays the anal-

ysis.

Zebra

This command creates a zebra plot of your surface. When you select the command the Zebra Plot Settings dialog box appears.



The dialog box contains the following elements:

Number of Stripes This data field contains the number stripes that appear on your surface.

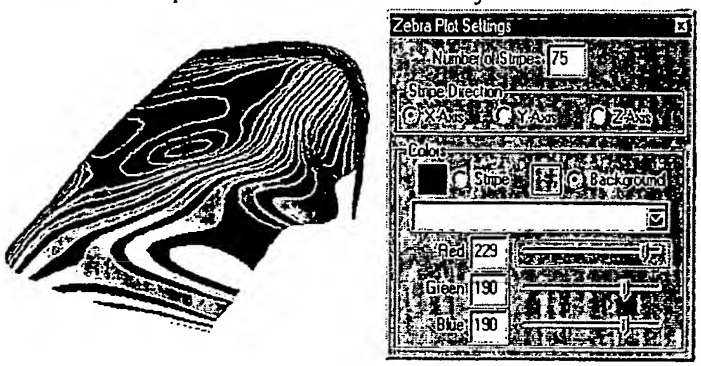
Stripe Direction This section provides three direction options: X-Axis, Y-Axis and Z-Axis.

Colors This sections contains the *Stripe Color* option and swatch, *Background Color* option and swatch, the color pull-down menu and RGB data fields and sliders.

Stripe Color option: When selected you can set the stripe color from the color pull-down menu or the RGB fields.

Background Color option: When selected you can set the surface background color from the color pull-down menu or the RGB fields.

This graphic shows an example of a zebra surface analysis.



Using the Zebra Command

1. Select the surface.
2. Choose ***Verify>Surface Analysis>Zebra***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.

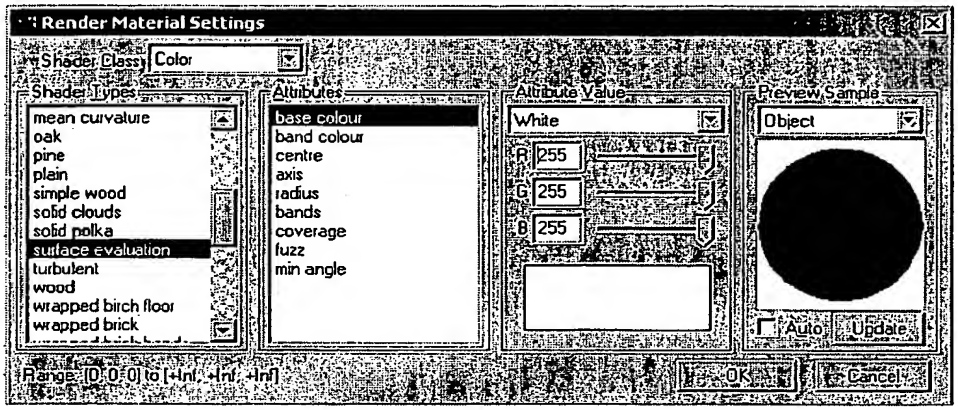
Evaluation Through the Advanced Rendering Feature

Through the advanced rendering feature you can evaluate your surfaces using user-defined attributes for the evaluation.

1. Select the surface.

   The surface must have a material applied to it.
2. Double-click on the surface to display the Edit Objects dialog box.
3. In the *Material* page of Edit Objects, click the Advanced button.

4. From the Color class choose the surface evaluation type. The following window displays:



base color This attribute sets the base color of the cylinder used for the evaluation. This color is reflected onto your surface. If your surface color is dark and your base color is white, you will not notice much change to the surface color when rendered.

band color This attribute sets the band or stripe color for the cylinder. This color is also reflected onto your surface. Use a color that is clearly visible to create the surface evaluation.

center This attribute sets the center of the evaluation cylinder. Click the desired location in your drawing and the numbers automatically enter into the X, Y and Z data fields.

axis This attribute specifies the cylinder axis (0 = X axis, 1 = Y axis, 2 = Z axis).

radius This attribute sets the cylinder radius.

bands This attribute sets the total number of bands around the evaluation cylinder.

coverage This attribute sets the ratio of the area covered by the bands to the area not covered (displaying the

	base color). Entering a zero displays only the base color. Entering a one displays only the band color.
fuzz	This attribute controls the fuzziness of the band (0 = sharp band edges, 1 = fuzzy band edges).
minimum angle	This attributes controls the angle of the normal along the cylinder axis that defines the length of the cylinder and thus the reflection on the surface. Smaller values create longer cylinders.

5. Specify the value for each attribute.
6. Click the Update button to preview your surface evaluation in the preview window.

   If you select the *Auto* option the preview window automatically updates each time you make an attribute change.
7. Click OK to save the changes and close the dialog box.

   Close the Edit Objects dialog box if you want a clean screen when you render your scene.
8. Render your scene using one of the photorealistic commands in the Render menu.

   The rendered scene shows your surface evaluation.

Solids Editing Tools - Features

After you've created a solid model from a primitive or a profile, you may use any of the feature tools to modify it. You cannot use these tools on grouped objects.

For each tool you can enter values in the Status Line either before or after you create the feature. If you enter the values after you select the tool but before you create the feature, your first click in the drawing area automatically registers all Status Line values. If you enter values after creating the feature and while it is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the feature to reflect the new values.

Many of the tools have a two step process to use them. The first step after choosing the tool may be to select one or more objects. The second step is to perform the editing operation. Once you the complete the first step (which may involve holding down the SHIFT key to select multiple objects), you cannot select more objects to include in the same operation. To select additional objects reset the tool and start again.

After adding features to your solid you can modify them by selecting them in the Design Explorer and choosing ***Window>Edit Objects*** or double-click on their names in the Design Explorer **(Cobalt™ and Xenon™ only)**.

All features are associative. Any modifications made to the parent will automatically alter all child objects that reference it **(Cobalt™ and Xenon™ only)**.

The topics explained here include:

- **Blend** tools

- **Chamfer Edge** tools
- **Hole** tools
- **Boss** Feature
- **Shell Object** tool
- **Bend Solid** tool
- **Feature Editing** tools

Blend Tools

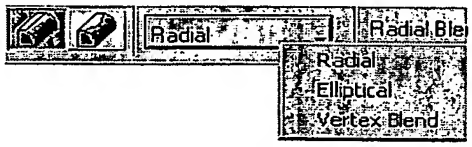
Tech Note:

The **Blend** tools performs a similar operation on solids to what the fillet tool performs on a wireframe.

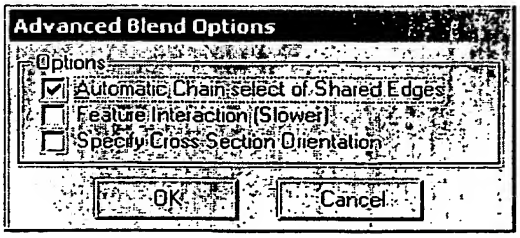
In the solids world, blend refers to both filleting and rounding. A filleting operation adds material to a concave intersection. A rounding operation removes material from a convex intersection.

The Blend tools create fillets along the edge of a solid. When you select the Blend tool, a subpalette appears in the Message Line containing two tools for creating blends: Constant Blend and Variable Blend.

Each of these tools includes a pull-down menu containing options for creating that type of blend. The graphic shown here is the Constant Length Blend tool menu.



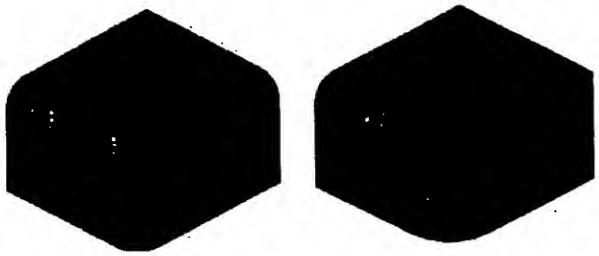
Many of these tool options also have advanced capabilities. These include: Constant Radial Blend, Constant Elliptical Blend, Variable Linear Blend, Variable By Position Blend, Variable Fixed Width Blend and Variable Hold Line Blend. These capabilities are indicated in the Message Line by the notation, [Ctrl = Advanced] (Windows) or [Option = Advanced] (Macintosh). When you press the CTRL (Windows) or OPTION (Macintosh) key, the Advanced Blend Options dialog box appears.



The dialog box contains the following options:

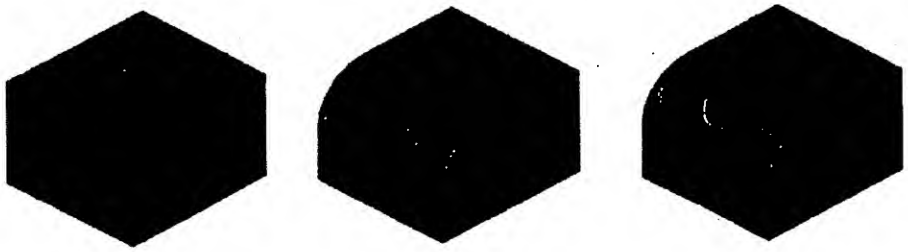
Automatic Chain-select of Shared Edges

Checking this box enables you to blend all edges that share a tangent with the selected edges. This is the default setting. Without this checked only the selected edges blend. The left graphic shows three edges blended with the Constant Radial Blend tool with this option selected. The right graphic shows two edges blended with the Constant Radial Blend tool without the option selected.



Feature Interaction (Slower)

Checking this box allows you to apply a blend that intersects a cutout or protrusion. Without this checked the feature may be deleted. With this checked the cutout is taken into account. The graphics here illustrate this. The left graphic is the original part with a hole cutout to half the part's depth. The middle right graphic is the blended part without this option selected. The right graphic is the blended part with the option selected.



The same thing occurs with a protrusion as shown below. As before, the left graphic is the original part;

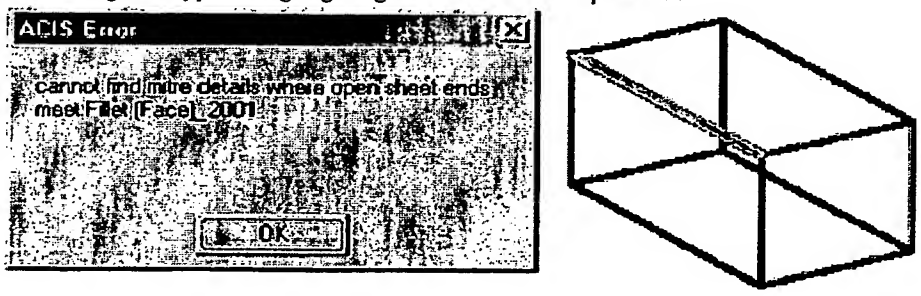
the middle graphic is the part without the option selected and the right graphic is the part with the option selected.

Using this option results in a longer blending time.

Specify Cross Section Orientation

Checking this box allows you to specify the orientation of the cross-section for the blend. This is especially valuable if you are blending an edge between faces containing a draft and you want to maintain the arc. See the next section for instruction on using this option.

If your model contains a problem that prevents a successful blending operation, an ACIS Error dialog box appears highlighting the location of the problem.

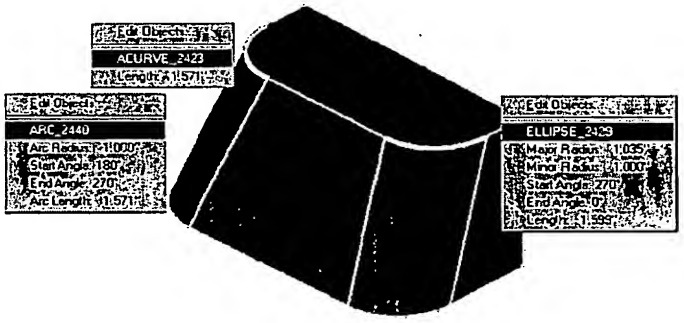


Specifying a Cross Section Orientation for a Blend

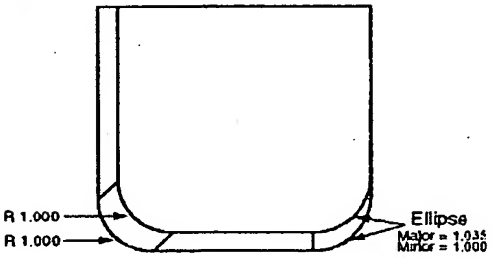
By default, this Designer Elements program uses the selected edge to define the orientation of the cross-section resulting in an arc shape for the blend. However, when you apply a blend to an object with a draft or taper, the cross-section of the resulting blend is elliptical because the blended edge is not perpendicular to the intersecting faces. If you want to

maintain the arc shape for the cross-section, use the Specify Cross Section Orientation option in the Advanced Blend Options dialog box.

The graphic below shows a tapered object (tapered on the left and front sides) with a cross-section blend on the left and a standard blend on the right. If you hold down the right mouse button (Windows) or CONTROL and the mouse button (Macintosh), you can verify the object shape. (In the case of the cross-section blend an acurve object is listed. Use one of the Arc tools to trace the shape and verify that the blend shape is an arc.)



The graphic to the right is a top view of a similar shape and provides the measurements for the blends.



Before using this option you must create a curve along the z axis for specifying the cross-section orientation.

Using the Cross Section Orientation Option

Although the example here uses the Radial option of the Constant Blend tool, this option is also available for Constant Elliptical Blends, Variable Linear Blends, Variable By Position Blends, Variable Fixed Width Blends and Variable Hold Line Blends.

1. Create a curve along the z-axis.
2. Select the Blend tool.

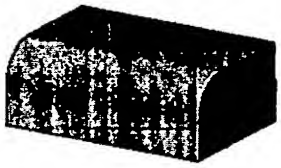
3. In the Message Line select the Constant Blend tool and the Radial option for the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*

4. Press CTRL (Windows) or OPTION (Macintosh) to display the Advanced Blend Options dialog box.
5. Select the Specify Cross Section Orientation option and click OK to close the dialog box.
6. Select the edge(s) for the blend.

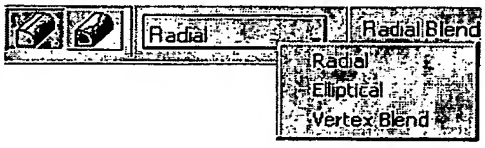
   The Message Line now reads, *Blend Solid: Pick curve to define cross section orientation.*
7. Select the curve.

   The blend is created.

Constant Blend Tool



When you select this tool its option pull-down menu appears.



You have three constant blend options: Radial, Elliptical and Vertex Blend.

Important: (Windows users) When a constant blend option uses the CTRL and SHIFT keys to perform specific operations for the tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when blending.

Tip:

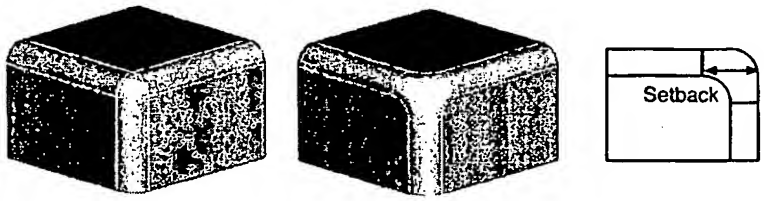
If you are having difficulty creating a radial blend, try creating a variable linear blend using the same radius for the beginning and the end. ACIS uses a different algorithm for linear blends.

Using the Radial Option

This option adds a blend with a constant radius along the edges you select and with a radius you specify in the Status Line.

If you select three or more edges you can create a smooth vertex blend or a setback blend. A setback is the distance that a blend extends back along the object edge from the vertex of the intersecting edges. A spherical corner results at the intersection of the three edges.

The left graphic below shows a three edge blend with no setback; the middle graphic shows the same object with a setback and the right graphic is a setback illustration.



1. Select the Blend tool.
2. In the Message Line select the Constant Blend tool and the Radial option for the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*

   Set the advanced blend options by pressing CTRL (Windows) or OPTION (Macintosh), if desired.
3. If you want a setback enter the value in the Status Line data field.
4. Select the edge(s) and/or face(s) for the blend.

While the blend is still selected you can change the radius of the blend in the Status Line. Type a new value and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Radius and Setback for the blend.

Geometric Characteristics

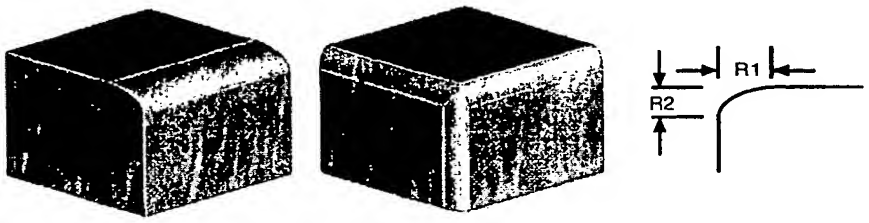
A constant radial blend is created by choosing a solid edge and specifying a radius and setback, if desired. It is made up of these characteristics according to Edit Objects: Setback, Radius and a list of the edges blended with their radii.

You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Using the Elliptical Option

This option adds a blend with a constant elliptical radius along the edges, based on the radii you specify in the Status Line. When using this option the automatic chain select of Shared Edges feature does not function. You must either select all intersecting edges

before blending or blend each edge separately. The left graphic below shows a basic elliptical blend. The middle graphic shows an elliptical blend with a mitered edge. The right graphic illustrates the radius variables.



1. Select the Blend tool.
2. In the Message Line select the Constant Blend tool and the Elliptical option from the pull-down menu. The Message Line reads: *Elliptical Blend: Pick edges to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend*].

   Set the advanced blend options by pressing CTRL (Windows) or OPTION (Macintosh), if desired.
3. Enter a radius values for the ellipse, R1 and R2, in the Status Line data field. R1 is the major axis radius and R2 is the minor axis radius.
4. Select the edge(s) for the blend.

While the blend is still selected you can change the radius and elliptical ratio of the blend in the Status Line. Type a new value and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the Radius and Ratio for the blend.

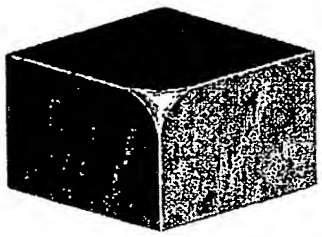
Geometric Characteristics

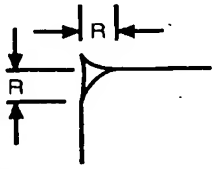
A constant elliptical blend is created by choosing a solid edge and specifying a radii for the major and minor axis of the ellipse. It is made up of these characteristics according to Edit Objects: R1, R2 and a list of the edges blended with their radii.

You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Using the Vertex Blend Option

This option adds a vertex blend at the intersection of three or more edges based on the radius you specify in the Status Line. The left graphic shows an example of a vertex blend. The right graphic illustrates the radius variables.



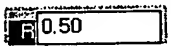


1. Select the Blend tool.

2. In the Message Line select the Constant Blend tool and the Vertex Blend option from the pull-down menu. The Message Line reads: *Vertex Fillet: Select vertex to blend.*
3. Enter a radius in the Status Line data field.
4. Select the vertex for the blend.

While the blend is still selected you can change the radius of the blend in the Status Line. Type a new value and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the radius (R) for the blend.



Geometric Characteristics

A constant vertex blend is created by choosing three or more solid edges and specifying a radius. It is made up of one characteristic according to Edit Objects: Radius.

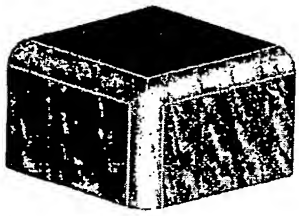
You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Additional Examples

The Constant Blend tool enables you to create a variety of blends. Here are some examples of advanced blends and the steps to create them.

Mitered Blend

A mitered blend can be created by turning off the Automatic Chain-select of Shared Edges option in the Advanced Blend Options dialog box and selecting each edge individually. A mitered corner results at the intersection of the three edges.



Creating a Mitered Blend

1. Select the Blend tool.
2. In the Message Line, select the Constant Blend tool and the Radial option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*
3. Press CTRL (Windows) or Option (Macintosh) to display the Advanced Blend Options dialog box.
4. Turn off the Automatic Chain-select of Shared Edges option and click OK.
5. Enter the desired radius in the Status Line.
6. Select each of the edges individually to create a mitered intersection. (If you hold down the SHIFT key a smooth vertex blend will be created rather than a mitered blend.)

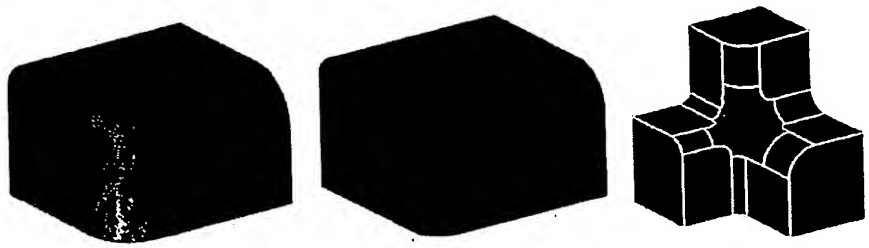
   If you want a different radius for each edge blend enter the new value in the Radius data field before selecting the edge.

   A mitered blend is created.

Multi-radius Blend without and with a Setback

A multi-radius blend is created after applying a constant radial blend to an object and then changing the radius of each edge through the Edit Objects dialog box. You can also add a setback to this blend through Edit Objects or when creating the radial blend initially. The

left graphic shows a multi-radius blend. The middle graphic shows the same object with a setback. The right graphic shows a multi-radius, multi-edge blend with a setback.



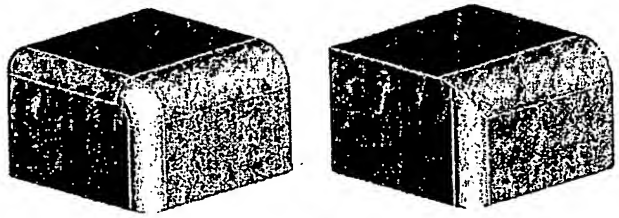
1. Select the object.
2. Choose ***Window>Design Explorer***.
3. Expand the history tree to display the fillet blend.(**Cobalt™** and **Xenon™** only)
4. Double-click on the fillet blend to display the Edit Objects dialog box.
5. Select the edge with the radius you want to change. The radius value appears in the data field. Enter a new radius.

   Change the other radii, if desired.
6. Click Apply to accept the changes and your geometry updates.

Chain Select Shared Edges On/Off Blend

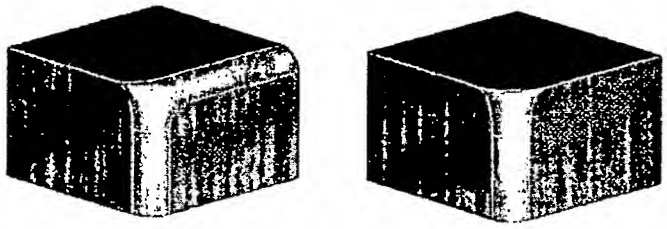
Two different kinds of blends are created depending on whether the Automatic Chain-select of Shared Edges option in the Advanced Blend Options dialog box is selected.

When this option is selected on edges with different radii, an object blends to create tangent continuous edges. In the left graphic the two edges have the same radius. When this option is not selected and different radii are used, the edges are capped, as shown in the right graphic.



Two Edge or One Edge Vertex Blend

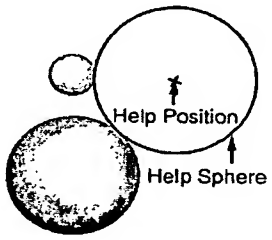
You can create two and one edge vertex blends by first creating a constant radial blend. Using the Design Explorer double-click on the blend to display the Edit Objects dialog box. For a two edge vertex blend change the radius of one edge to zero. For a one edge vertex blend, change two radii to zero. The left graphic below shows the two edge vertex blend and the right graphic a one edge vertex blend.



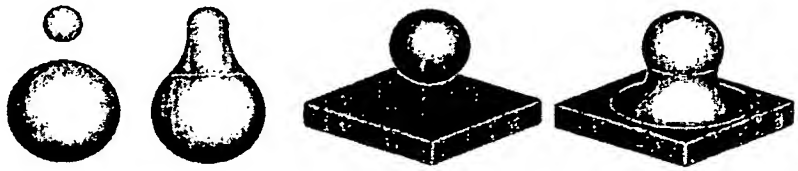
Disjoint Blend

A disjoint blend is a blend between faces that do not touch or are on two separate bodies. To create this blend in the past you had to perform a boolean operation. A disjoint blend is created by specifying a radius for the blend, selecting the two faces or objects and specifying a blend help position.

The help position tells this Designer Elements program where to begin calculating the blend. Visualize a sphere with this help position at its center. This sphere rolls along the virtual blend between the two bodies as this Designer Elements program calculates the actual blend. The sphere diameter equals twice the blend radius. The illustration here shows the help sphere and the help position for two disjoint bodies and applies to all disjoint blend operations.



The graphics below are two examples of objects before and after a disjoint blend is created.



Tips:

- If a help position does not create the blend try another location in the same quadrant.
- When creating a blend between objects of different colors, the color of the first object selected becomes the color of the new blended object.

Creating a Disjoint Blend

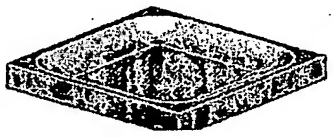
1. Select the Blend tool.
2. In the Message Line, select the Constant Blend tool and the Radial option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*
3. Hold the SHIFT key down and select the objects. The Message Line now reads, *Constant Blend: Specify help position.*
4. Click the help position location. (If the help position does not create the blend try another location in the same quadrant.)

   A disjoint blend is created.

Disjoint Edge to Face Blend

A disjoint edge to face blend is created by selecting the edge and face of an object. What makes this blend unique is that our Designer Elements program allows you to specify a radius for the blend that is greater than the distance from the edge to the face.

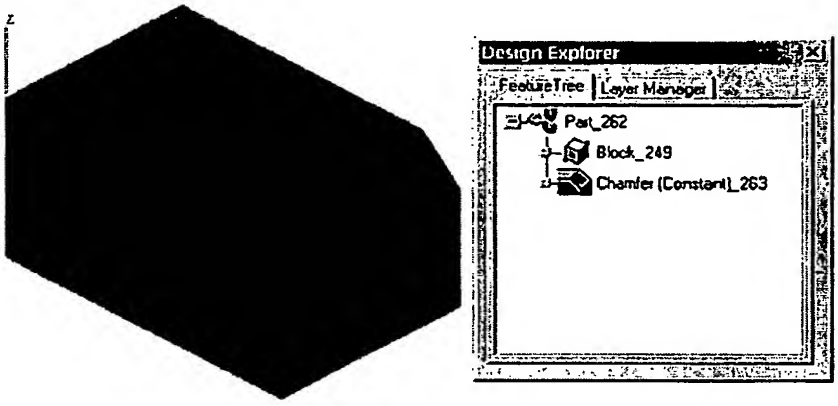
In the graphic here a one inch blend is applied using the inside top edge of the object and its inside bottom face. The user supplied help position is important in this blend. If you choose the wrong location the blend will not work. The one inch radius is larger than the distance from the edge to the face.



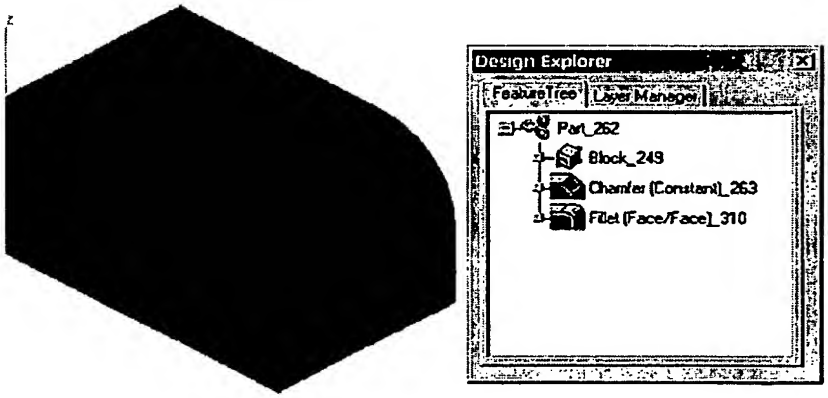
Disjoint Face to Face Remote Blend

A disjoint face to face remote blend is created when applying a blend to two faces that do not share the same edge. In graphic below, a face to face remote blend is applied to the top and right face separated by a chamfer. Below are the before (chamfered block) and after pictures (Blended top and right face seperated by the chamfer).

Before.

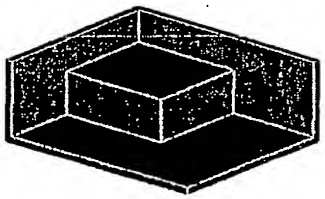


After



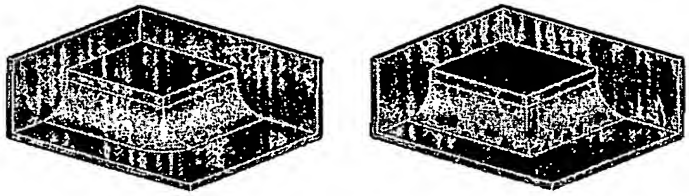
Notice in the Design Explorer, shown above, that the fillet blend does not replace the chamfer but follows the chamfer operation in the history tree.

Edge to Face Roll Blend

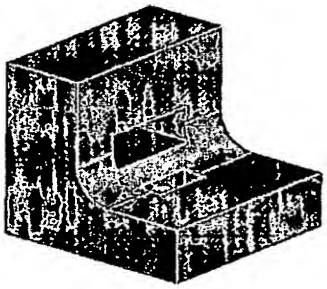


An edge to face roll blend is create by specifying a radius, selecting an edge and face to blend and choosing a help position. The graphic to the right shows the original object used for this blend.

The left graphic below shows the edge to face roll blend in which the vertical edge and the inside bottom face are selected. Using the Drafting Assistant, the help position is located out from the shared edge/face corner vertex along the z axis. The right graphic shows the default blend if the vertical faces and inside bottom faces were selected.



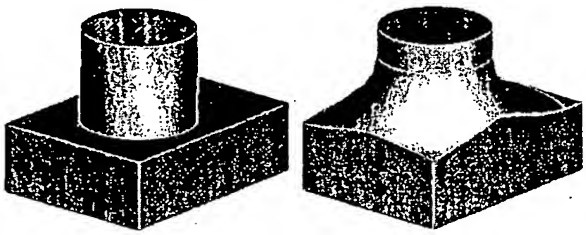
Blend with Cutouts



A blend with cutouts is created by applying a blend to faces that contain cutouts using the Constant Blend tool. There are no special steps to create this blend.

Extrapolation Blend

An extrapolation blend is created when you apply a constant radial blend to an edge with a radius that is larger than what the intersecting face will accommodate. The intersecting face extends to accept the blend. The left graphic below is the original object. The right graphic shows that the faces were extended to accommodate the blend.

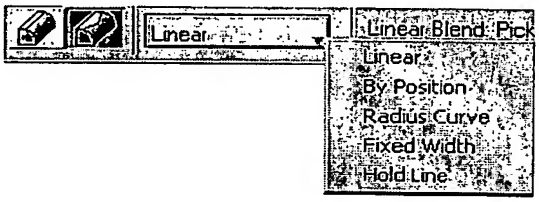


Creating an Extrapolation Blend

1. Select the Blend tool.
2. In the Message Line, select the Constant Blend tool and the Radial option from the pull-down menu. The Message Line reads: *Radial Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*
3. Enter the radius in the Status Line.
4. Select the edge to blend.

   The blend is created and the intersecting side extends as necessary.

Variable Blend Tool



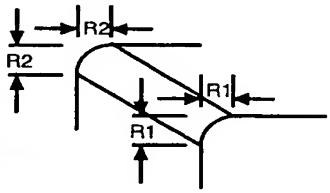
When you select this tool its option pull-down menu appears.

You have five variable blend options: Linear, By Position, Radius Curve, Fixed Width and Hold Line.

Important: (Windows users) When a variable blend option uses the CTRL and SHIFT keys to perform specific operations for the tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when blending.

Using the Linear Option

This option adds a blend with a tapered radius along the edges you select. You specify the beginning radius (R1) and the ending radius (R2) in the Status Line.



1. Select the Blend tool.
2. In the Message Line, select the Variable Blend tool and the Linear option from the pull-down menu. The Message Line reads: *Linear Blend: Pick edge to linearly blend. [Pick edge near R1] [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh)].*

3. Select the edge along which to make the blend.

   The solid edge closest to your point of selection becomes R1.

While the blend is still selected you can change the radii (R1 and R2) in the Message Line. Type the new values and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains R1 (beginning radius) and R2 (ending radius) for the linear blend.

Geometric Characteristics

A variable linear blend is created by choosing a solid edge and specifying a radius for the beginning and end of the blend. It is made up of these characteristics according to Edit Objects: Radius 1 and Radius 2.

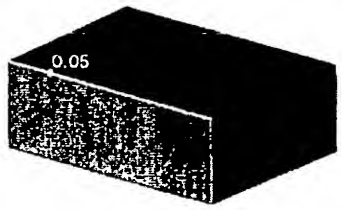
You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Using the By Position Option

This option adds a blend with a varied radius along the edges you select. You specify the various radii in the Status Line.

1. Select the Blend tool.
2. In the Message Line, select the Variable Blend tool and the By Position option from the pull-down menu. The Message Line reads: *Position Blend: Pick edge(s) for variable radius blend. [End=ESC,Dbl Click] [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh)] [Shift=Extend].*
3. Select the edge along which to make the blend. The Message Line reads: *Position Blend: Specify location for radius value [End=ESC, Dbl Click].*
4. Enter a radius in the R field of the Status Line.
5. Specify the location(s) along the selected edge for the radius.

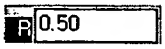
As you apply a radius to a location the value appears on the edge.



6. Enter a new radius in the R field.
7. Specify the location for this radius along the edge.

Continue following this procedure until you have entered all the desired radii. Double-click the last place radius point and the variable by position blend is created.

The Status Line contains the Radius for the blend.



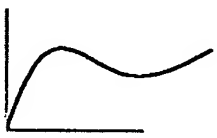
Geometric Characteristics

A variable position blend is created by choosing a solid edge and various radii at different locations along the edge for the blend. It is made up of these characteristics according to Edit Objects: a list of the edges blended with their radii and Param points, Radius and Param value. Param is the percentage point location along the edge for the selected item in the list. When you select one of the items in the list its Radius displays in the Radius data field and its percentage point location displays in the Param field.

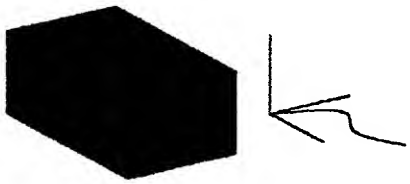
You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Parameters command from the Design Explorer menu displays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

Using the Radius Curve Option

This option adds a blend based on a specified radius curve. Before using this option you must create an open curve in the positive xy quadrant. This curve cannot cross over itself or backtrack. A circle or a loop would not be acceptable curves. The graphic here is an example of an acceptable curve.

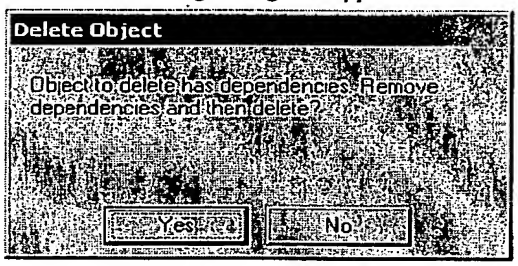


This curve defines the radius distribution. The start point of the curve represents the start point of the edge. The y value of the curve represents the radius. The x length of the curve is divided into percentages. The y value at a particular per-



centage point on the curve is mapped as the radius to the same percentage point along the selected solid edge.

One advantage of this tool is the ability to modify the blend by changing the curve. You can also add or remove control points to the radius curve to change the blend. The blend is associative to the curve. Changing the curve, automatically changes the blend. If you attempt to delete the curve, the following dialog box appears:



Deleting the curve changes the object from a solid block to an ACIS solid.

1. Create the curve that will be referenced for the blend.

   To ensure that the curve is in the positive xy quadrant, change the view to Top and draw the curve using the Axis as a reference.

1. Select the Blend tool.
2. In the Message Line, select the Variable Blend tool and the Radius Curve option from the pull-down menu. The Message Line reads: *Radius Curve Blend: Pick edge for variable radius blend.*
3. Select the solid edge. The Message Line now reads, *Radius Curve Blend: Pick radius curve (in positive xy plane).*
4. Select the radius curve.

   The blend is applied to the edge.

There are no Status Line entries.

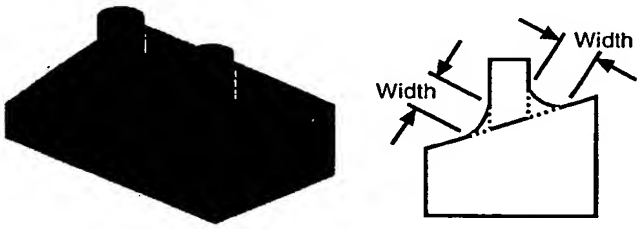
Geometric Characteristics

A variable radius curve blend is created by choosing a solid edge and a curve in the positive xy quadrant to specify the radii along the edge. According to Edit Objects, a radius curve blend contains no unique geometric characteristics. You can make changes to the Display and Attribute settings through the Edit Objects dialog box. You cannot change the blend radius through the Edit Objects dialog box. It must be done using the original curve.

Select the curve and change the x or y location in Edit Objects or drag the control point on the curve to the new positive x, or y location (The z location must remain at zero).

Using the Fixed Width Option

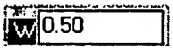
This option adds a blend with a fixed width along the edges you select adjusting the blend radius to maintain the width. This is important when placing a blend along the edge of two faces where the angle changes. You specify the width in the Status Line. In the graphic, the left boss has a constant radial blend with a .5 inch radius and the right graphic a fixed width blend with a .5 inch width.



1. Select the Blend tool.
2. In the Message Line, select the Variable Blend tool and the Fixed Width option from the pull-down menu. The Message Line reads: *Fixed Width Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*
3. Enter the fixed width for the blend in the Status Line.
4. Select the edge for the blend.

   A fixed width blend is created.

The Status Line contains the width (W) of the blend.



Geometric Characteristics

A variable fixed width blend is created by choosing a solid edge and specifying the blend width. It is made up of these characteristics according to Edit Objects: Radius and a list of edges blended with their associated width.

You can select the blend using the Design Explorer. Double-clicking on the item in the Design Explorer or choosing Edit Objects command from the Design Explorer menu dis-

plays the Edit Objects dialog box. You can make changes to the settings on the Display and Attribute pages. See Chapter 24 for more information on the Design Explorer.

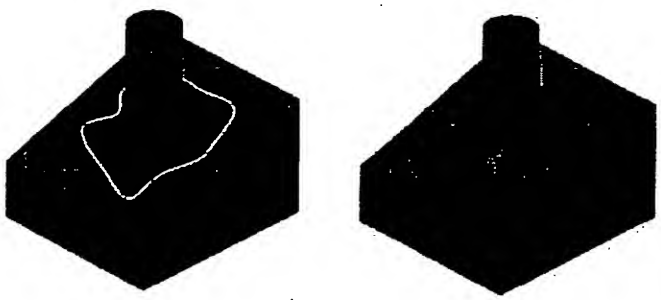
Using the Hold Line Option

This option adds a blend to the hold line curve specified adjusting the blend radius as needed to fit the line.

Hold Line Rules:

- The hold line curve should be one curve (circle, ellipse or spline). If the curve is not closed, a linear blend is applied where there is no hold line using the open ends of the curve for the radius values.
- Although the hold line curve does not have to be on the plane of the surface, it should be close to it and in the same plane as the surface. Curves not on the surface are projected to the closest surface to calculate the blend.
- Grouped curves are not supported.

The left graphic shows a part with a spline on the slanted face. The right graphic shows the same part using the spline as the hold line for the blend.



1. Create a hold line curve.
2. Select the Blend tool.
3. In the Message Line, select the Variable Blend tool and the Hold Line option from the pull-down menu. The Message Line reads: *Hold Line Blend: Pick edges AND/OR faces to blend [Ctrl=Advanced (Windows) or Option=Advanced (Macintosh), Shift=Extend].*

4. Select the edge for the blend. The Message Line now reads, *Hold Line Blend: Pick curve for hold line [Shift = Extend].*

5. Select the hold line curve.

   A hold line blend is created.

There are no Status Line entries.

Geometric Characteristics

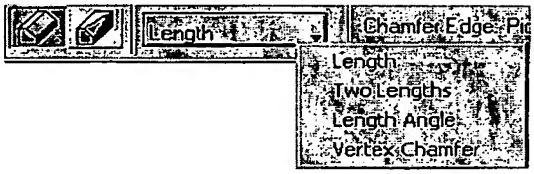
A variable hold line blend is created by choosing a solid edge and a curve for the hold line. According to the Edit Objects, a hold line blend contains no unique geometric characteristics. You can make changes to the Display and Attribute settings through the Edit Objects dialog box.

Chamfer Edge Tools

The Chamfer Edge tools create flat surfaces along a specified edge. The graphic here shows a chamfered solid along all four edges.

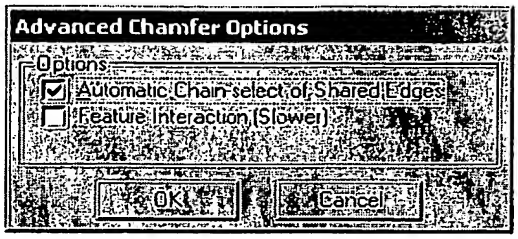
When you select the Chamfer Edge tool a subpalette appears in the Message Line containing two chamfer edge tools: Constant Length and Variable Length.

Each of these tools includes a pull-down menu containing options for creating that type of chamfer. The graphic shown here is the Constant Length Chamfer tool menu.



Many of these tool options also have advanced capabilities. These include: Constant Length Chamfer, Two Lengths Chamfer, Length Angle Chamfer, Variable Lengths Chamfer, Variable Four Lengths Chamfer, Variable Lengths Angle Chamfer and Variable By Position Chamfer. These capabilities are indicated in the Message Line by the notation, [Ctrl = Advanced] (Windows) or [Option =

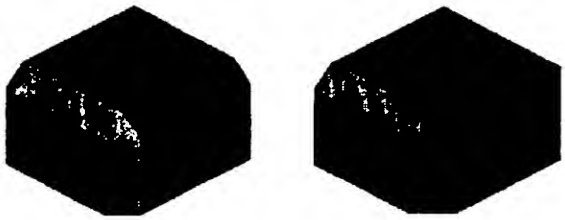
Advanced] (Macintosh). When you press the CTRL (Windows) or OPTION (Macintosh) key, the Advanced Chamfer Options dialog box appears.



The dialog box contains the following options:

Automatic Chain-select of Shared Edges

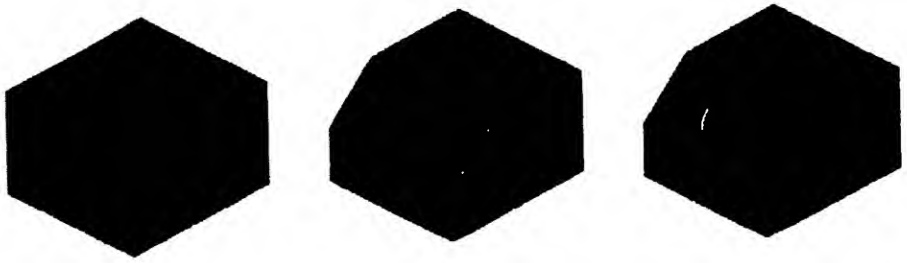
Checking this box enables you to chamfer all edges that share a tangent with the selected edges. This is the default setting. Without this checked only the selected edges chamfer. The left graphic shows three edges chamfered with the Constant Length Chamfer tool with the option selected. The right graphic shows two edges chamfered with the Constant Length Chamfer tool without the option selected.



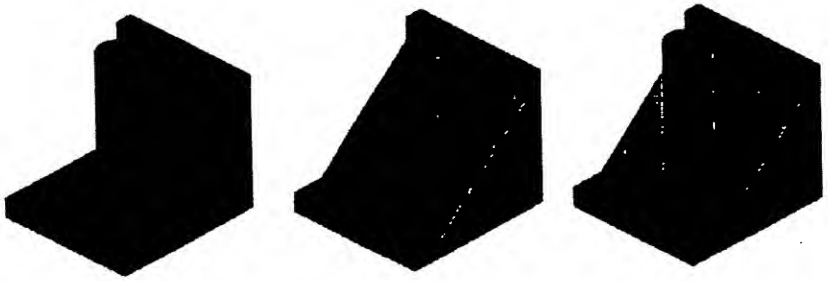
Feature Interaction (Slower)

Checking this box allows you to apply a chamfer that intersects a cutout or protrusion. Without this checked the feature may be deleted. With it checked the cutout is taken into account. The graphics illustrate this. The left graphic is the original part with hole cutout to half the part's depth. The middle graphic is the chamfered part

without this option selected. The right graphic is the chamfered part with the option selected.



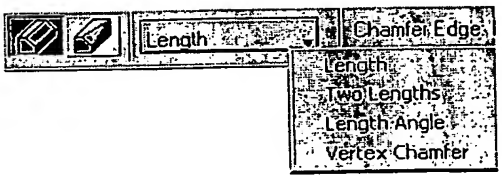
Without this checked for a protrusion the chamfer takes precedence over the protrusion. The left graphic below is the original part. The middle graphic is the part without the option selected. The right graphic is the part with the option selected.



Using this option results in a longer chamfering time.

Constant Length Chamfer Tool

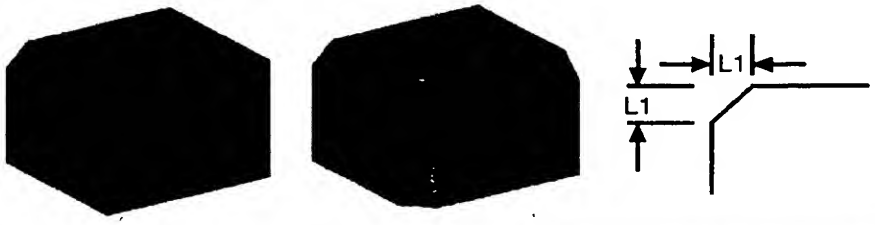
When you select this tool its option pull-down menu appears.



You have four constant length options: Length, Two Lengths, Length Angle and Vertex Chamfer.

Using the Length Option

This option adds a 45° chamfer based on the specified length to edge distance along the selected edge. The left graphic is an example of an object using this option. The middle graphic here shows a .5 inch chamfer with a vertex setback of 1.0. (You create this by selecting three or more edges with a setback at the corner vertex.) The right graphic illustrates the variables.



1. Select the Chamfer Edge tool.

2. In the Message Line, select the Constant Length Chamfer tool and the Length option from the pull-down menu. The Message Line reads: *Chamfer Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*
3. Enter a setback value in the Setback data field of the Status Line, if desired, for three or more intersecting edges.
4. Select the edge(s) to chamfer.

   The chamfer is created.

While the chamfer is still selected you can change L1 and the setback length in the Status Line. Type the desired values in the data field and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1 and the Setback.

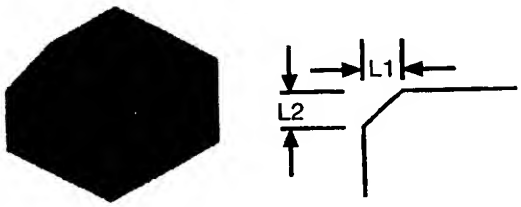
Geometric Characteristics

A constant length chamfer is created by choosing a solid edge and specifying the length to the edge. It is made up of two characteristic according to Edit Objects: Setback and Length. The setback is the distance from the shared vertex of three or more edges that the chamfer extends.

It also includes an edge list displaying the edge name and its associate length. To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Two Lengths Option

Using this option adds a chamfer based on the specified length to edge distances along the selected edge to create a constant length chamfer.



1. Select the Chamfer Edge tool.
2. In the Message Line, select the Constant Length Chamfer tool and the Two Lengths option from the pull-down menu. The Message Line reads: *Chamfer Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*
3. Enter L1 and L2 values in the Status Line.
4. Select the edge(s) to chamfer.

   The chamfer is created.

While the chamfer is still selected you can change the L1 and L2 values in the Status Line. Type the desired values in the data field and press ENTER (Windows) or RETURN (Macintosh).

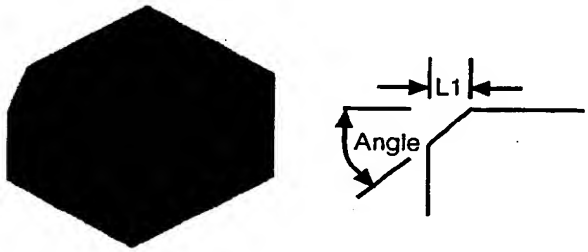
The Status Line contains L1 and L2.

Geometric Characteristics

A constant two length chamfer is created by choosing a solid edge and specifying the lengths to the edge. It is made up of two characteristics according to Edit Objects: Length 1 and Length 2. To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Length Angle Option

This option adds a chamfer at an angle and a constant length to the edge that you specify in the Status Line. The angle is measured from the tangent of the selected edge to the chamfered edge (see the right graphic).



1. Select the Chamfer Edge tool.
2. In the Message Line, select the Constant Length Chamfer tool and the Length Angle option from the pull-down menu. The Message Line reads: *Chamfer Angle Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*
3. Enter an L1 and angle in the Status Line data fields.
4. Select the edge(s) to chamfer.

While the chamfer is still selected you can change the length and angle in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1 and the A chamfer angle) data fields.

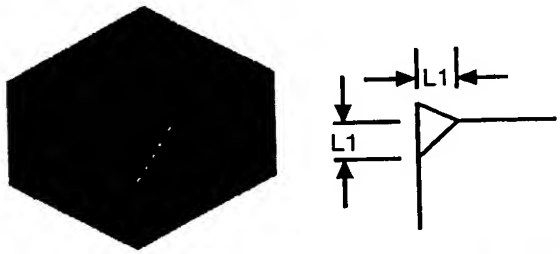
Geometric Characteristics

An constant length angle chamfer is created by choosing a solid edge and specifying an angle and length. It is made up of these characteristics according to Edit Objects: Setback and Angle. The setback is the distance from the shared vertex of three or more edges that the chamfer extends.

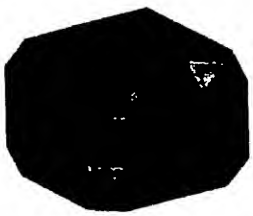
To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Vertex Chamfer Option

This option adds a constant vertex chamfer to the intersection of three edges. The length is measured from the vertex to the edge as indicated in the right graphic below.



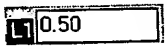
The faces next to the intersecting edges must be planar and have the same convexity or concavity. The graphic here shows convex and concave vertex chamfers.



1. Select the Chamfer Edge tool.
2. In the Message Line select the Constant Length Chamfer tool and the Vertex Chamfer option from the pull-down menu. The Message Line reads: *Vertex Corner: Select vertex to corner.*
3. Enter an L1 value in the Status Line data field.
4. Select the vertex to chamfer.

While the chamfer is still selected you can change the length in the Status Line. Type the desired value in the data field and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1 chamfer length.



Geometric Characteristics

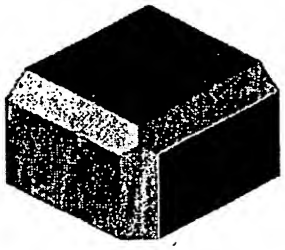
An vertex corner chamfer is created by choosing a vertex and specifying a length. It is made up of one characteristic according to Edit Objects: Radius 1. To modify the chamfer radius select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Additional Examples

Here are a few chamfer variations using the constant chamfer tools.

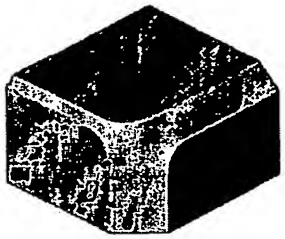
Multi-edge Chamfer with Mitered Corner

This chamfer is created using the Length option of the Constant Length Chamfer tool. In the Advanced Chamfer options dialog box turn off the Automatic Chain-select of Shared Edges option and then chamfer each edge separately.



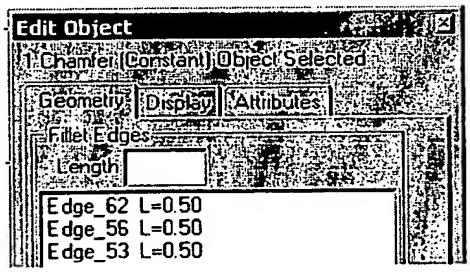
Multi-edge Chamfer with Varied Lengths and Setback

This chamfer is created using the Length option of the Constant Length Chamfer tool.



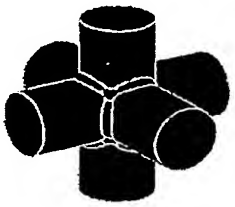
1. Select the Chamfer Edge tool.

2. In the Message Line, select the Constant Length Chamfer tool and the Length option from the pull-down menu.
   The Message Line reads: *Chamfer Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*
3. Enter a setback value in the Setback data field.
4. Hold down the SHIFT key and select the edges to chamfer them.
5. Using the Design Explorer, double-click on the chamfer to display the Edit Object dialog box.
6. In the edge list select an edge. It's associated L1 value appears in the length data field.
7. Enter a new value in the field.
   Change other values as desired.
8. Click apply and the chamfer updates.



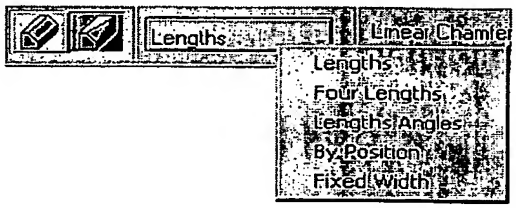
Complex Chamfer

This Designer Elements program supports complex chamfering for intersecting objects with or without setbacks. The example here includes setbacks.



Variable Chamfer Edge Tool

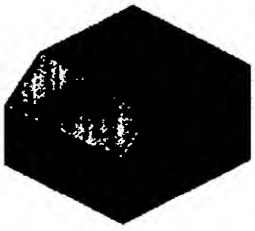
When you select this tool its option pull-down menu appears.

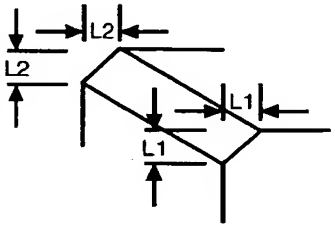


You have five variable length options: Lengths, Four Lengths, Lengths Angles, By Position and Fixed Width.

Using the Lengths Option

This option adds a chamfer tapered linearly along the edge from two specified setback lengths, L1 and L2.





1. Select the Chamfer Edge tool.

2. In the Message Line, select the Variable Length Chamfer tool and the Lengths option from the pull-down menu. The Message Line reads: *Linear Chamfer: Pick edge to linearly chamfer. (Pick edge near L1) [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)].*
3. Enter the L1 (the beginning setback length of the chamfer) and L2 (the ending setback length of the chamfer) values in the Status Line data fields.
4. Select the edge to chamfer.

While the chamfer is still selected you can change the setback lengths (L1 and L2) in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the chamfer setback lengths, L1 and L2.

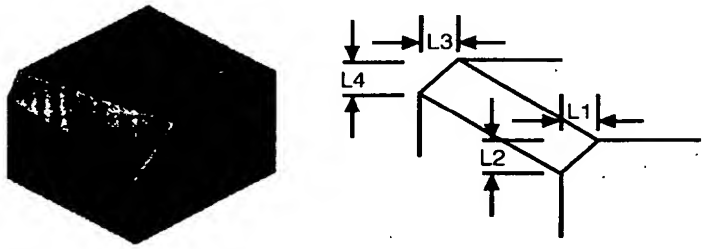
Geometric Characteristics

A variable lengths or linear chamfer is created by choosing a solid edge and specifying a length for the beginning and end of the chamfer. It is made up of these characteristics according to Edit Objects: L1, L2 and a list of the edges chamfered with their corresponding lengths.

To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Four Lengths Option

This option adds a chamfer tapered linearly along the edge using four specified setback lengths, L1, L2, L3 and L4.



1. Select the Chamfer Edge tool.
2. In the Message Line select the Variable Length Chamfer tool and the Four Lengths option from the pull-down menu. In the Message Line select the Linear Chamfer tool. The Message Line reads: *Chamfer Edge: Pick solid edges to chamfer. [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*
3. Enter the L1 and L2 values for the beginning setback lengths of the chamfer and L3 and L4 values for the ending setback lengths of the chamfer in the Status Line data fields.
4. Select the edge(s) to chamfer.

While the chamfer is still selected you can change the setback lengths in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the chamfer setback lengths, L1, L2, L3 and L4.

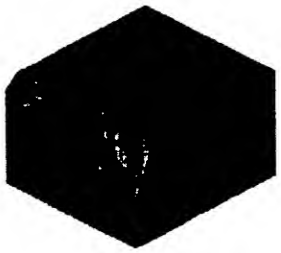
Geometric Characteristics

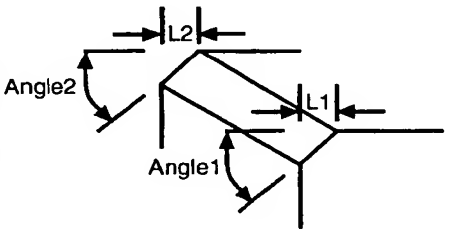
A variable four lengths chamfer is created by choosing a solid edge and specifying a lengths for the beginning and end of the chamfer. It is made up of these characteristics according to Edit Objects: L1, L2, L3, L4 and a list of the edges chamfered with their corresponding lengths.

To modify the chamfer characteristics select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Variable Lengths Angles Option

This option adds a chamfer with the specified lengths and angles for the beginning and end of the chamfer along the selected edge. The angles are measured from the tangent of the selected edge to the chamfered edge (see the right graphic).





1. Select the Chamfer Edge tool.
2. In the Message Line, select the Variable Length Chamfer tool and the Lengths Angles option from the pull-down menu. The Message Line reads: *Chamfer Angle Edge: Pick solid edges to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*
3. Enter an L1 and A1 (angle) for the beginning of the chamfer and L2 and A2 (angle) for the end of the chamfer in the Status Line data fields.

4. Select the edge(s) to chamfer.

While the chamfer is still selected, you can change the lengths and angles in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains L1, A1, L2 and A2 data fields.

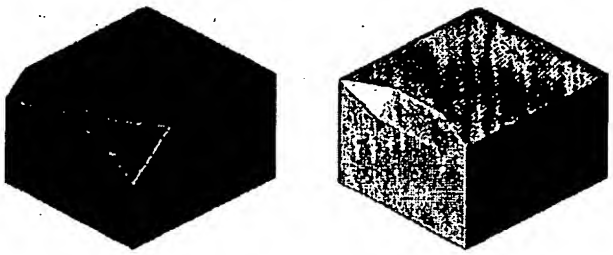
Geometric Characteristics

A variable length angle chamfer is created by choosing a solid edge and specifying the beginning and end chamfer angles and lengths. It is made up of these characteristics according to Edit Objects: L1, Angle 1, L2, Angle 2 and a list of the edges chamfered with their corresponding lengths and angles.

To modify the chamfer characteristics, select the chamfer using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the chamfer name. See Chapter 24 for more information on the Design Explorer.

Using the Variable By Position Option

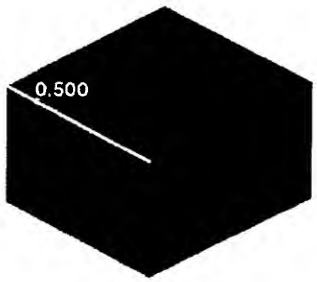
This option adds a chamfer with a varied lengths along the edges you select. You specify the lengths in the Status Line. You can also create a degenerative chamfer with this option by specifying a zero length at both ends of the edge. The left graphic shows a varied length chamfer. The right graphic shows a degenerative chamfer.



1. Select the Chamfer Edge tool.
2. In the Message Line, select the Variable Length Chamfer tool and the By Position option from the pull-down menu. The Message Line reads: *Position Chamfer: Pick edges for variable position chamfer [End = ESC, Dbl Click] [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*

3. Select the edge along which to make the chamfer. The Message Line reads: *Position Chamfer: Specify location for length value [End=ESC, Dbl Click].*
4. Enter a setback length in the Length field of the Status Line.
5. Specify the location(s) along the selected edge for the chamfer.

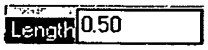
   As you specify the location the length appears in red at that point and all additional locations. The values remain until you double-click to specify the last chamfer point and the chamfer is created. The graphic illustrates this.



6. Enter a new length in the Length field.
7. Specify the location for this length along the edge.

   Continue following this procedure until you have entered all the desired lengths. Double-click the last place chamfer point and the variable by position chamfer is created.

The Status Line contains the Length for the chamfer setback.



Geometric Characteristics

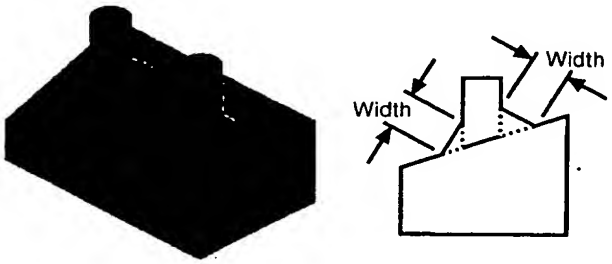
A variable by position chamfer is created by choosing a solid edge and various setback lengths at different locations along the edge for the chamfer. It is made up of these characteristics according to Edit Objects: a list of the edges chamfered with their widths and Param points, Width (which corresponds to the Status Line length) and Param value. Param is the percentage point location along the edge for the selected item in the list. When you select one of the items in the list its Length displays in the Width data field and its percentage point location displays in the Param field.

To modify the chamfer characteristics select the chamfer using the Design Explorer and choose ***Window>Edit Objects***. See Chapter 24 for more information on the Design Explorer.

Using the Fixed Width Option

This option adds a chamfer with a fixed width along the edges you select adjusting the chamfer length to maintain the width. This is important when placing a chamfer along of the edge of two faces where the angle changes. You specify the width in the Status Line. The left graphic shows a sloped block with two bosses where the left boss has at constant

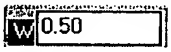
length chamfer and the right boss has a variable fixed width chamfer. The right graphic illustrates the fixed width characteristic.



1. Select the Chamfer Edge tool.
2. In the Message Line select the Variable Length Chamfer tool and the Fixed Width option from the pull-down menu. The Message Line reads: *Fixed Width Chamfer: Pick edges AND/OR faces to chamfer [Ctrl = Advanced (Windows) or Option = Advanced (Macintosh)] [Shift=Extend].*
3. Enter a width in the W data field of the Status Line.
4. Pick the edge(s) for the chamfer.

While the chamfer is still selected you can change the width in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the width for the chamfer.



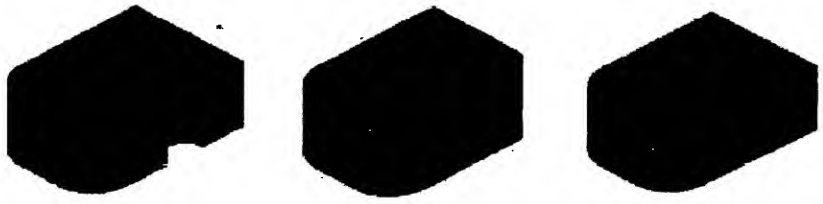
Geometric Characteristics

A variable fixed width chamfer is created by setting a chamfer width and selecting the edge for the chamfer. It is made up of these characteristics according to Edit Objects: Width and a list of the edges chamfered with their width. When you select one of the items in the list its width displays in the Width data field.

To modify the chamfer characteristics select the chamfer using the Design Explorer and choose ***Window>Edit Objects***. See Chapter 24 for more information on the Design Explorer.

Hole Feature Tools

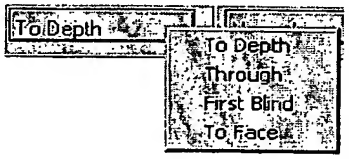
With the Hole Feature tools you can model bolt holes found frequently in mechanical design parts. This Designer Elements program holes are predefined features that remove cylindrical shapes from objects. Holes can be placed through models, along face edges at corner vertices and on planar and non-planar faces. The graphic below is an example of a simple hole.



Holes, like other features in this Designer Elements program, are associative. You can modify them at any point in your design process. When you select the Hole Feature tools a subpalette appears containing three tools for creating holes: Simple, Counter Bore and Counter Sink.

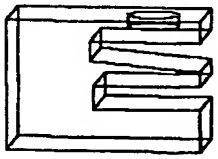
Through Types

The Message Line also contains a pull-down menu allowing you to choose a Through Type for your hole. You have four options: Depth, Through, First Blind and To Face.



Depth

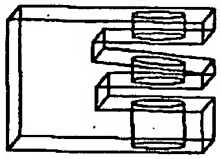
Extends the depth of the hole to the value specified in the Status Line.



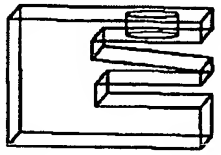
Tech Note:

If you want to place a hole through an object that will intersect a blend, use the *Depth* option rather than the *To Face* option. If *To Face* is used, when the hole intersects the blend, a face change occurs and the hole ends.

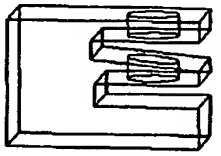
Through Extends a hole through the selected solid.



First Blind Extends a hole to the first open face.

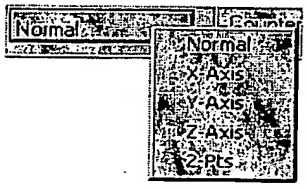


To Face Extends a hole to the selected face.



Hole Direction

The Message Line contains a pull-down menu allowing you to specify a boss direction. You have five options: Normal, X-Axis, Y-Axis, Z-Axis and 2-Pts.



Normal Creates a hole perpendicular to the solid face.

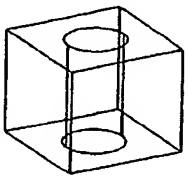
X-Axis Creates a hole along the X-axis.

Y-Axis Creates a hole along the Y-axis.

Z-Axis Creates a hole along the Z-axis.

2-Pts Creates a hole in the direction specified by two user defined points.

Simple Hole Tool



This tool creates straight, cylindrical holes through the selected solid. You can specify the X, Y and Z center location, depth, diameter and draft angle of the hole in the Status Line.

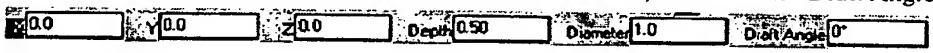
Using the Simple Hole Tool

1. Select the Hole tool.
2. In the Message Line select the Simple Hole tool. The Message Line reads: *Simple Hole: Pick solid face to place hole.*
3. Select a Through Type and Hole Direction from their respective pull-down menus in the Message line.
4. Select the face where you want to locate the hole.
5. Select the location for the center of the hole.
6. If you selected the 2-Pts direction click two points to specify the direction.

   The hole is created.

While the solid is still selected you can change the hole depth in the Status Line. Type the desired value in the data field and press ENTER (Windows) or RETURN (Macintosh).

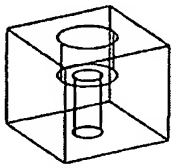
The Status Line contains the hole's X, Y and Z center location, Diameter and Draft Angle.



Geometric Characteristics

A simple hole is created by choosing the solid face and the location of the hole. It is made up of these characteristics according to Edit Objects: Hole Type, Thru Type, Hole Depth, Diameter and Draft Angle. To modify the hole characteristics, select the hole using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the hole name. See Chapter 24 for more information on the Design Explorer.

Counter Bore Tool



This tool creates a counter bore hole based on the values you specify in the Status Line. A counter bore consists of the bore and the hole. The bore is a straight-sided cylinder with a diameter larger than the hole. The hole extends from the end of the bore and completes the counter bore. See the graphic here.

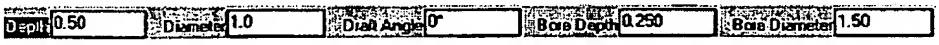
Using the Counter Bore Hole Tool

1. Select the Hole tool.
2. In the Message Line select the Counter Bore Hole tool. The Message Line reads: *Counter Bore: Pick solid face to place counterbore.*
3. Select a Through Type and Hole Direction from their respective pull-down menus in the Message Line.
4. Select the face to place the counter bore.
5. Specify the location for the center of the counterbore hole.
6. If you selected the 2-Pts direction click two points to specify the direction.

   The bore is created.

While the solid is still selected you can change the depth, diameter and draft angle of the hole and the bore depth and radius in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

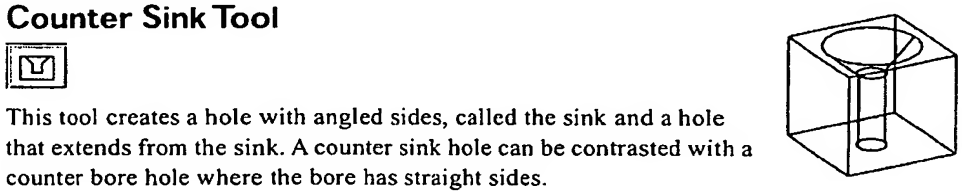
The Status Line contains the counter bore's Depth, Diameter, Draft Angle, Bore Depth and Bore Diameter.



Geometric Characteristics

A counter bore is created by choosing the solid face and the location of the hole. It is made up of these characteristics according to Edit Objects: Hole Type, Thru Type, Hole Depth, Diameter, Draft Angle, Counter Bore Diameter and Depth. To modify the hole characteristics, select the hole using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the hole name. See Chapter 24 for more information on the Design Explorer.

Counter Sink Tool



This tool creates a hole with angled sides, called the sink and a hole that extends from the sink. A counter sink hole can be contrasted with a counter bore hole where the bore has straight sides.

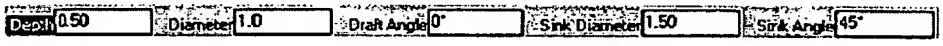
Using the Counter Sink Hole Tool

1. Select the Hole tool.
2. In the Message Line select the Counter Sink Hole tool. The Message Line reads: *Counter Sink Hole: Pick solid face to place countersink.*
3. Select a Through Type and Hole Direction from their respective pull-down menus in the Message line.
4. Select the face where you want to locate the hole.
5. Select the location for the center of the hole.
6. If you selected the 2-Pts direction click two points to specify the direction.

   The hole is created.

While the solid is still selected you can change the depth, diameter and draft angle, sink radius and sink angle in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the countersink's Depth, Diameter, Draft Angle, Sink Diameter and Sink Angle.

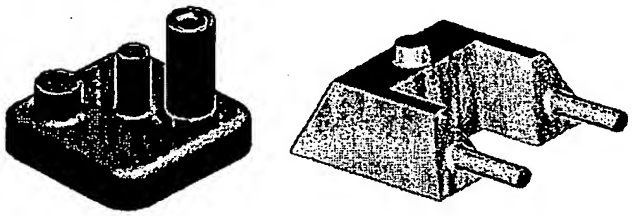


Geometric Characteristics

A counter sink hole is created by choosing the solid face and the location of the hole. It is made up of these characteristics according to Edit Objects: Hole Type, Thru Type, Hole Depth, Diameter, Draft Angle, Counter Sink Diameter and Angle. To modify the hole characteristics, select the hole using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the hole name. See Chapter 24 for more information on the Design Explorer.

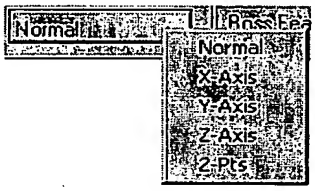
Boss Feature Tool

This tool creates a boss by adding a cylinder of a specified size to another solid and filleting the intersection of the two. A boss can be placed on planar and non-planar surfaces. You specify the height, diameter and fillet radius in the Status Line.



Boss Direction

The Message Line contains a pull-down menu allowing you to specify a boss direction. You have five options: Normal, X-Axis, Y-Axis, Z-Axis and 2-Pts.



Normal	Creates a boss perpendicular to the solid face.
X-Axis	Creates a boss along the X-axis.
Y-Axis	Creates a boss along the Y-axis.
Z-Axis	Creates a boss along the Z-axis.
2-Pts	Creates a boss in the direction specified by two user defined points.

Using the Boss Feature Tool

1. Select the Boss Feature tool. The Message Line reads: *Boss Feature: Pick solid face to place boss.*

2. Select the boss direction from the pull-down menu.

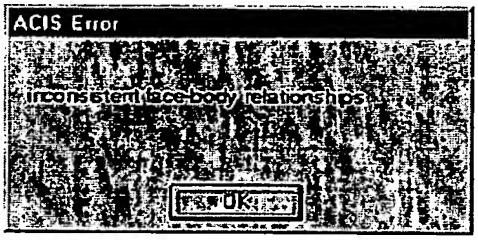
3. Select the face to locate the boss.
4. Specify a location for the boss center.
5. If you selected the 2-Pts direction click two points to specify the direction.

   The boss is created.

While the solid is still selected you can change the height, diameter and fillet radius of the boss in the Status Line. Type the desired values in the data fields and press ENTER (Windows) or RETURN (Macintosh).

The Status Line contains the boss' Height, Radius and Fillet Radius.

If the solid face selected in incompatible with the axis direction the following ACIS Error appears.



Geometric Characteristics

A boss is created by choosing the solid face and specifying the values for the cylinder. It is made up of these characteristics according to Edit Objects: Height, Diameter and Fillet Radius. To modify the boss characteristics, select the boss using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the boss name. See Chapter 24 for more information on the Design Explorer.

Feature Editing

One of this Designer Elements program' more powerful features is the transparent parent/child associativity it creates for solid objects. As you add features to your part this Designer Elements program automatically establishes relationships between them. When you modify or change a parent, this Designer Elements program updates and regenerates all the children. For example, consider the following procedure:

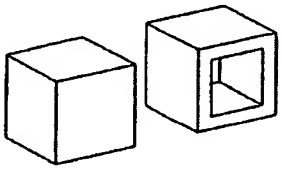
1. Create a block primitive.
2. Fillet the block's edges.
3. Place a boss at the block's center.

4. Place a countersink hole at the boss' top center.

Now try these modifications:

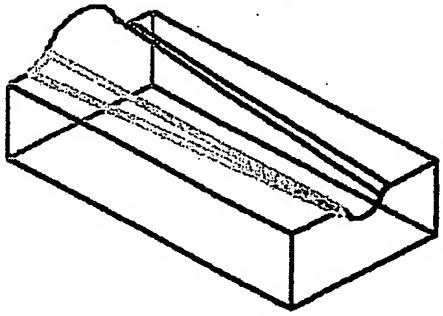
5. Select a face or edge from the original block. Choose ***Window>Edit Objects*** and change the block's length, width, and height. The fillets, boss and countersink hole automatically regenerate based on the new length, width, and height.
6. Select the block. Choose ***Window>Design Explorer***. Click on the + sign to display the history tree of the block. Double-click on the fillet name in the Design Explorer. The Edit Objects dialog box appears. Change the fillet's radius. Note that the boss and hole automatically regenerate.
7. In the Design Explorer select one of the boss' face names. The Edit Object dialog box now displays the boss information. Change the boss to 100000 (or any outrageous value). In this case this Designer Elements program cannot regenerate the solid and returns it to the previously valid state.

Shell Solid Tool



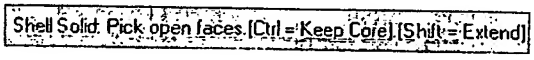
The Shell Solid tool hollows out a solid with a thickness you set.

This tool also includes an error detection routine. If your model contains a problem that prevents the completion of the local face modeling operation, an ACIS error dialog box appears with the problem location highlighted.



After shelling you can edit the face thickness in the Edit Objects dialog box.

This tool also gives you the option of keeping the core of the shelled object. Notice the Messages Line:



Using the Shell Solid Tool

1. Select the Shell Solid tool. The Message Line reads: *Shell Solid: Pick solid to shell.*

2. In the Status Line, enter the desired offset or thickness of the shell.
3. Select the solid. The Message Line reads: *Shell Solid: Pick open faces [Shift=Extend].*
4. Select the face or faces that will be open. The object shells.

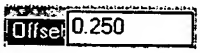
   If you click in an area with no faces, the object will be hollowed out with no open faces.

Tech Note:

Like all tools, you can set the thickness of the shell after the operation using the Status Line. When working with small objects, you may need to change the default thickness first to shell successfully.

While the solid is still selected you can enter a new value in the offset field. Press ENTER (Windows) or RETURN (Macintosh) to accept the change.

The Status Line contains the offset or shell thickness.



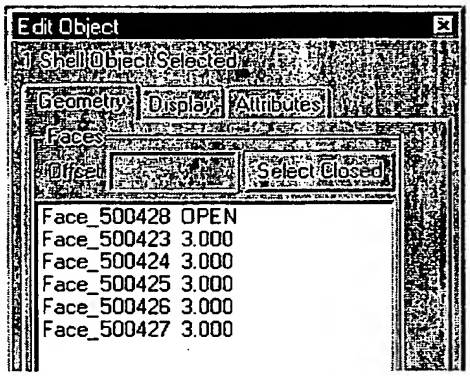
Modifying a Shelled Solid

After you shell a solid you can modify the shelling operation through the Edit Objects dialog box.

1. Select the shelled solid.
2. Choose ***Window>Design Explorer.***
3. Open the history tree for the solid by clicking on the + (plus sign).
4. Select the shell operation.

5. Choose ***Window>Edit Objects.***

   Edit Objects includes the Offset data field, Face list and a Selection button which toggles between three operations, Select Open, Select Closed and Select All.

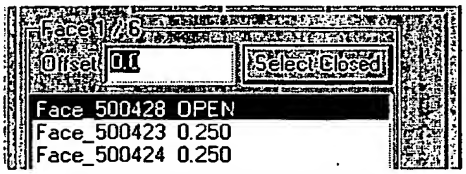


Offset This field displays the offset when a shell face is selected from the face list.

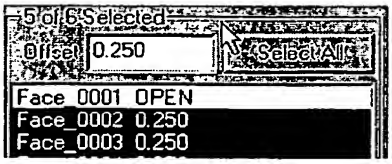
Face list This list specifies the faces contained in the solid, a Designer Elements identifier number and the offset for the face. If the face was removed in place of an offset value appears the word open. If you suppress a face, suppressed is displayed instead of an offset value.

Select Open This selection operation displays when all faces are selected. Click this button to select all open faces.

Select Closed This selection operation displays when an open face or no faces are selected. Click this button to select all closed faces.



Select All This selection operation displays when all closed faces are selected. Click this button to select all faces.



If you select one or more faces manually the buttons will not toggle automatically. You can also change the more than one face by holding down the SHIFT key when selecting or dragging to select multiple faces.

6. Select the desired face. (As you move through the list of faces, the selected face is highlighted on your object.)
7. Make any desired modifications.

   You can suppress a face or change the offset for each face to another value. An offset of 0 removes the face.
8. Click Apply to accept the changes.

Geometric Characteristics

A shelled solid is created by choosing a solid, specifying an offset and the face(s) you want open. It is made up of these characteristics according to Edit Objects: Offset thickness, Suppress Face check box and a list of the solid faces with the respective offset distance. This list is only available if you select the operation in the Design Explorer first and then display the Edit Object dialog box. See Chapter 24 for more information on the Design Explorer.

Bend Solid Tool

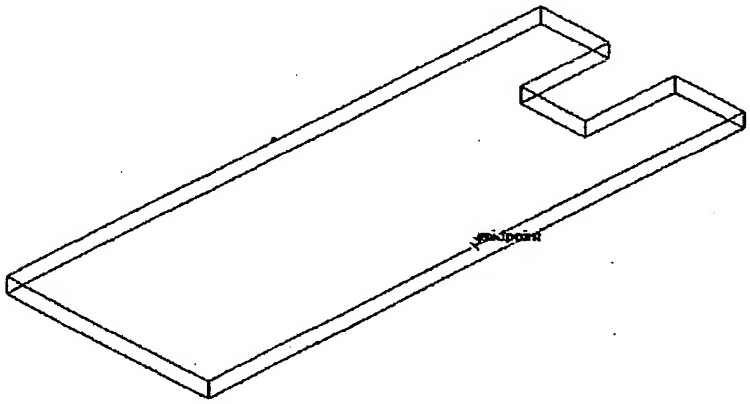
The Bend Solid tool creates a solid by bending a solid about an axis through a given radius.

There are three options in the Message Line; Bend, Center Bend and Bend Along Curve.

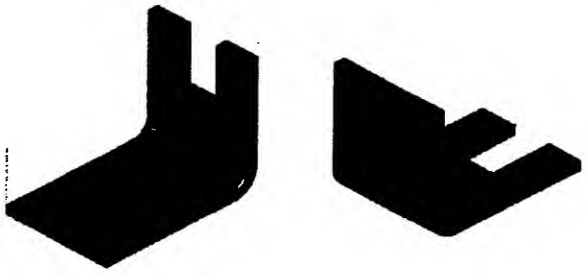
Using the Bend Axis Option

1. Click the Bend Solid tool. Select the first tool in the Message Line. The Message Line reads: *Bend Solid: Pick Solid to Bend.*

2. The Status Line contains the Bend Radius and Bend Angle fields. Enter the desired values for your bend solid. Tab between data fields.
3. Pick the solid that you wish to bend
4. The message line now reads, *Bend Solid: Enter two points for bend axis.* Specify two points for the bend axis. The two points represent the inner tangent line for the bend solid. After you click the two points, the solid is bent about the axis.



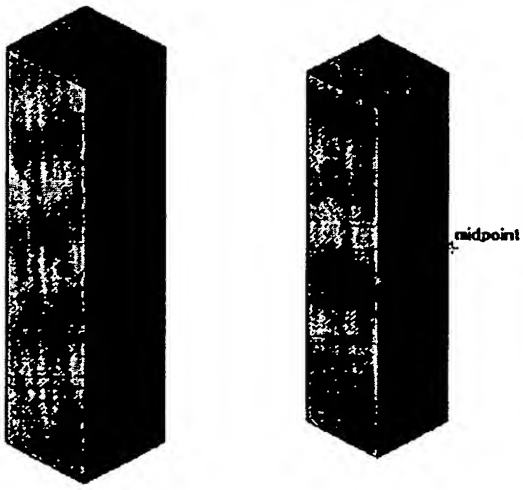
5. The Message Line now reads, *Bend Solid: Pick Solid to Bend [Option = Flip Material].* Hit the option key to flip the side of the material that was bent.



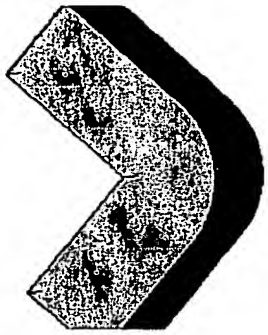
While the object is still selected, you can edit the bend radius and bend angle. Enter the new values and press ENTER (Windows) or RETURN (Macintosh) and the solid updates

Using the Center Bend Option

1. Click the Bend Solid tool. Select the first tool in the Message Line. The Message Line reads: *Center Bend Solid: Pick Solid to Bend.*
2. The Status Line contains the Bend Radius and Bend Angle fields. Enter the desired values for your bend solid. Tab between data fields.
3. Pick the solid that you wish to bend
4. The message line now reads, *Bend Solid: Enter two points for bend axis.* Specify two points for the bend axis. The two points represent the inner tangent line for the bend. After you click the two points, the solid is bent about the axis.
5. The Message Line now reads, *Center Bend Solid: Pick Solid to Bend [Ctrl or Option = Flip Material].* Hit the option key to flip the side of the material that was bent.



Solid to Bend Bend Axis Resulting Bend

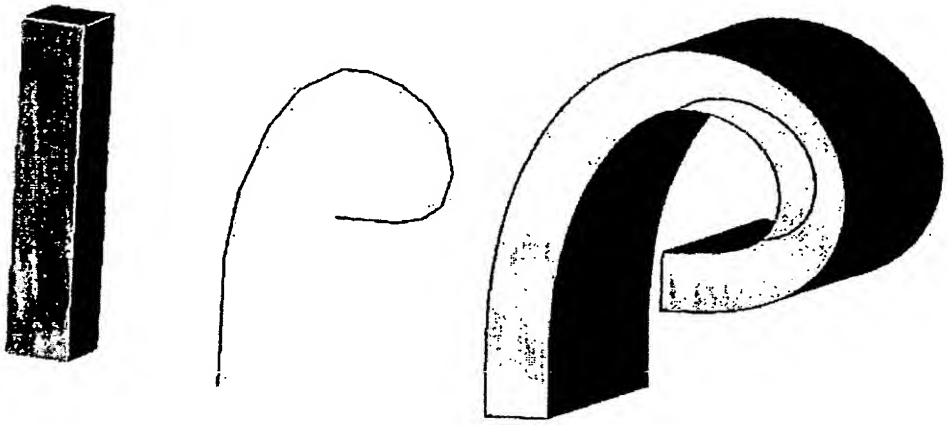


Flipped Material

Using the Bend Along Curve Option

1. Click the Bend Solid tool. Select the first tool in the Message Line. The Message Line reads: *Bend Solid Along Path: Pick Solid to Bend..*
2. Pick the solid that you wish to bend
3. The message line now reads, *Bend Solid Along Path: Pick curve to bend solid to.* Choose your curve.
4. The Message Line now reads, *Bend Solid: Enter two points for bend height axis.* Specify two points for the bend axis. The two points represent thepoint at which the bend will start.

5. The Message Line now reads, *Bend Solid: Enter two points for alignment axis.* Choose the points on the solid you want aligned with the curve.



Solid Utilities and Features

After you've created a solid model you may need to modify it. This Designer Elements program provides several advanced editing tools to accomplish this. Use Boolean routines to add solids to each other, subtract them or create a new solid from their intersections. With the other tools you can split a solid in two, stitch several surfaces together to form a solid, hollow out a solid to create a shell, change a solid face's draft angle or collectively change the draft angles of a set of upper and lower surfaces. You can also evaluate drafts placed on solid objects using the Draft Evaluation feature.

This Designer Elements program also provides you with assembly modeling tools for connecting, mating, aligning and inserts solids.

For each tool you can enter values in the Status Line to perform the editing operation either before or after the operation is complete. If you enter the values after you select the tool but before you perform the operation, your first click in the drawing area automatically registers all Status Line values. If you enter values after performing the operation and while the solid is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the solid to reflect the new values.

After using one of these tools to modify your solid you can make further changes to them by selecting the tool operation in the Design Explorer and choosing ***Window>Edit Objects.***

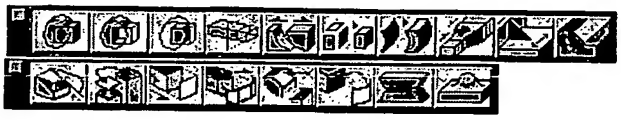
All editing operations conducted on a solid are associative. Any modifications made to the parent will automatically alter all child objects that reference it **(Cobalt™ and Xenon™ only)**.

The topics covered in this chapter include:

- **Solid Editing** tools
- Draft Evaluation feature
- **Assembly Modeling** tools

Solid Utility Tools

This Designer Elements program provides you with advanced editing or utilities tools located in the last subpalette of the Solids tool palette. The tools include: Boolean, Split Solid, Stitched Solid, Shell Solid, Parting Line Solid, Local Face Modeling, Thicken Solid and Deform Face.



Boolean Tools

Your Designer Elements programs **Boolean** tools allow you to perform boolean operations on solid objects. You can add solids together, subtract one solid from another or intersect a solid with others.

Tech Note:

The add, subtract and intersect tool replace the two original solids with a new composite solid. The only way to recover the original solids after this boolean operation is to undo the command. A technique for retaining the originals is to copy them to another layer before using a Boolean tool.

Union Solid Tool

The **Union Solid** tool combines two or more solids into one. In the left graphic below Solid A and Solid B are individual solids. The right graphic is a single solid.

Using the Union Solid Tool

1. Select the **Boolean** tool.

2. In the Message Line select the Union Solid tool. The Message Line reads: *Union Solid: Pick solid to add to and solid(s) to add [Shift=Extend].*
3. Select the solid to which you want to add.
4. Select the solid(s) that you want to add to the first solid. Hold down the SHIFT key before you select the first solid to select multiple solids.

There are no Status Line entries.

Subtract Solid Tool

The Subtract Solid tool subtracts one or more solids from another. In the right graphic Solid B is subtracted from Solid A.

Using the Subtract Solid Tool

1. Select the Boolean tool.
2. In the Message Line select the Subtract Solid tool. The Message Line reads: *Subtract Solid: Pick solid to subtract from and solid(s) to subtract out [Shift=Extend].*
3. Select the solid from which you want to subtract.
4. Select the solid(s) that you want to subtract from the first solid. Hold down the SHIFT key before you select the first solid to select multiple solids.

There are no Status Line entries.

Intersect Solid Tool

The Intersect Solid tool creates a new solid from the common volume of two intersecting solids. The right graphic below is the intersection of Solid A and Solid B.

Using the Intersect Solid Tool

1. Select the Boolean tool.
2. In the Message Line select the **Intersect Solid** tool. The Message Line reads: *Intersect Solid: Pick two solids to intersect [Shift=Extend].*
3. Select the two solids to form the intersect solid.

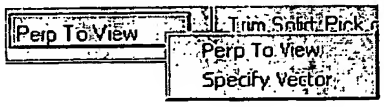
There are no Status Line entries.

Geometric Characteristics

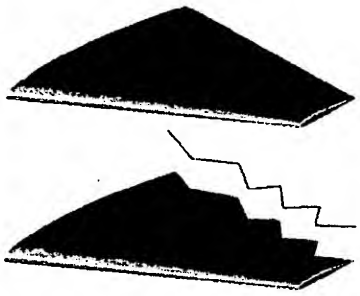
There are no characteristics listed in Edit Objects specific to a boolean solid.

Trim Solid Tool

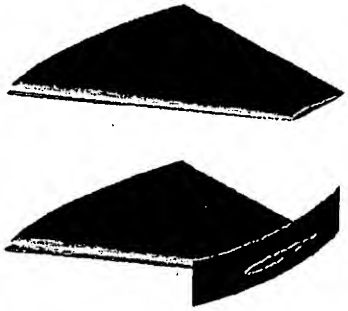
The Trim Solid tool trims a solid with a curve, surface, or solid. The trim tool is useful for removing material from a base solid. In addition to trimming to surfaces or another solid, the trim solid tool has the unique ability to allow trimming a solid to a curve. In the case of a curve, an internal surface is built that sweeps into the screen or sweeps by user defined vector. Use the pull down menu to specify your preference.



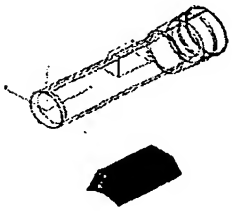
In the case of trimming a solid to a surface or other solid, the pull down option has no bearing on the results.



Trim Solid to Curve.



Trim Solid to Surface

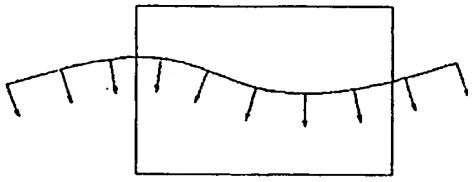


Trim Solid to Solid.

Using the Trim Solid Tool

1. Click the Trim Solid tool. The Message Line reads: *Trim Solid: Pick curve, surface or solid to trim to.*

2. The Message Line now reads, *Trim Solid: Pick solid to trim [Crtl (Windows) or Option (Macintosh) = Flip Direction to Remove*|



3. The solid is trimmed to the curve, removing the solid in the direction of the arrows.

4. Press the Control or Option key if you want to change the material that is kept.

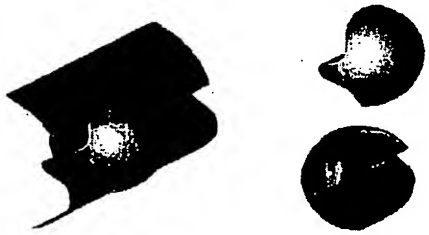
Note: Press the Control or Option key to change the direction of the material to be removed **after** the trim is completed. Do not hold the Control or Option keys during the trimming process.

When trimming a solid to another solid the portion that is kept is the portion that was selected by the user.

There is no Status Line for the trim solid tool.

Split Solid Tool

The Split Solid tool divides a solid into two parts. This Designer Elements program uses an infinite plane, surface, or solid as the splitting entity so you must construct the splitting object before you can make the split.



Using the Split Solid Tool

1. Select the **Split Solid** tool. The Message Line reads: *Split Solid: Pick solid to split.*

2. Select the solid. The Message Line reads: *Split Solid: Pick surface for splitting solid.*
3. Select the infinite plane or surface to split the solid.

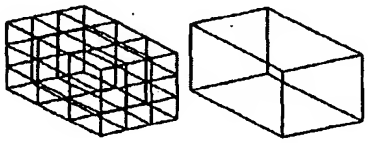
There are no Status Line entries.

Geometric Characteristics

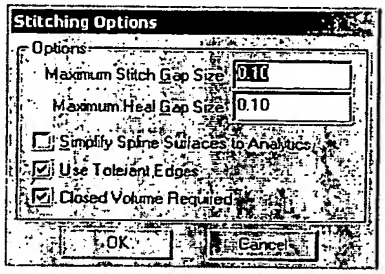
There are no characteristics listed in Edit Objects specific to a split solid.

Stitched Solid Tool

The **Stitched Solid** tool creates a closed solid from a collection of surfaces. The color and resolution of the first surface selected are used for the stitched solid.



The tool uses healing technology to repair any small gaps in the model. and creates a solid only if the selected surfaces create a closed body. This tool also provides you with three stitching options. Press the CTRL key (Windows) or OPTION key (Macintosh) to display the Stitching Options dialog box.



The dialog box contains the following options:

Maximum Heal Gap Size

This field set the maximum size that Vellum will attempt to stitch to objects. Enter a value between 0 and 1.

Example: If the gap size is .1 but the object has a gap of .2, the surfaces won't stitch.

Simplify Spline Surfaces to Analytics

This option sets whether spline surfaces are simplified when they are stitched. Solids use surfaces as their underlying geometry. These surfaces can be either Analytic surfaces or B-Spline Nurb surfaces. Analytic surfaces are used to create cylinders, cones and spheres. B-Splines surfaces are used to create a variety of shapes. ACIS is "turbocharged" for analytic surfaces.

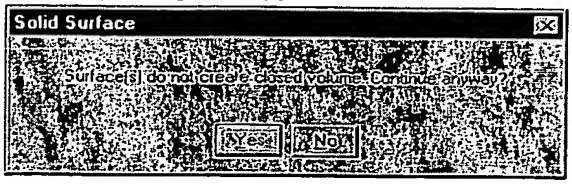
When selected, this Designer Elements program attempts to simplify your spline surfaces. If your surfaces create a standard shape such as a cylinder, they convert into analytic surfaces. This conversion speeds up the calculation process for the solid and for your file. By default, this option is not selected.

Use Tolerant Edges

This option determines whether tolerant edges will be used if a model can't stitch because of a gap. When selected and an edge can't stitch, it is marked as tolerant. Only operations that involve the entire edge can be conducted.

If this option is not selected and this Designer Elements program discovers a gap larger than speci-

fied in the Maximum Heal Gap Size data field the following dialog box appears.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when stitching solids.

Using the Stitched Solid Tool

1. Select the **Stitched Solid** tool. The Message Line reads: *Stitched Solid: Pick surfaces to create solid from. [Ctrl=Advanced]* (Windows) or *Option=Advanced* (Macintosh) *[Shift=Extend].*
2. Press the CTRL key (Windows) or OPTION key (Macintosh) to display the Stitching Options dialog box.
3. Specify the settings and click OK. Click Cancel to close the dialog box.
4. Hold down the SHIFT key and select the surfaces or drag a selection fence around them.

   The surfaces are stitched into a solid.

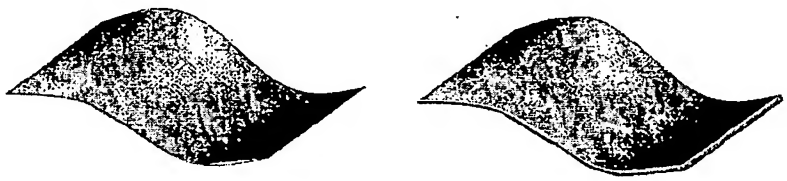
There are no Status Line entries.

Geometric Characteristics

There are no characteristics listed in Edit Objects specific to a stitched solid.

Thicken Solid Tool

The Thicken Solid tool allows you to thicken surfaces and solids. You specify the desired thickness. A positive or negative value determines the direction of the thickening. The left graphic shows a surface and the right graphic shows the thickened surface, now a solid.



Using the Thicken Solid Tool

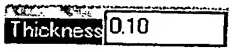
1. Select the Thicken Solid tool. The Message Line reads: *Thicken Solid: Pick surface or solid to thicken. [Ctrl or Option =Flip thickness side].*

2. Type the desired thickness in the Thickness data field in the Status Line.
3. Select the surface or solid object to be thickened. This Designer Elements program thickens the selected object.

By pressing the CONTROL or OPTION key you can toggle which side of the surface is thickened

While the object is still selected you can enter a new value in the Thickness data field and press ENTER (Windows) or RETURN (Macintosh) to change the thickness.

The Status Lines contains the Thickness value for the thickened object.



Geometric Characteristics

A thickened solid object is created by specifying a thickness and selecting the object. A thickened object is made up of the Thickness characteristic according to the Edit Objects dialog box. To modify the characteristics of the thickened object

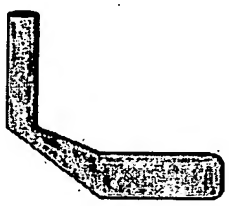
select the object using the Design Explorer (**Cobalt™ and Xenon™ only**) and choose ***Window>Edit Objects*** or double-click on the face name.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

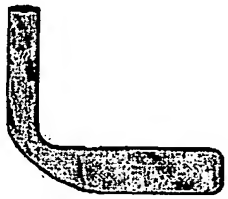
See Chapter 24 for more information on the Design Explorer.

Lofted Solid Tool

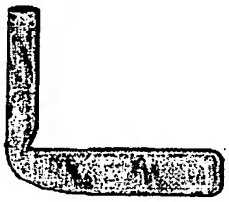
This tool creates a tangent solid between two neighboring solids using a specified bulge factor. The higher the bugle factor the more influential the slope of the tangent for the selected solid. A loft solid with a bulge factor of zero (0) creates a discontinuous loft. The graphics below illustrate the effects of the bulge factor.



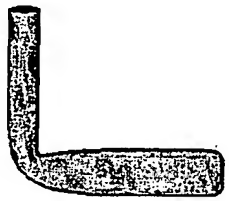
Bulge Factor = 0



Bulge Factor = 5



Bulge Factor 1 = 5; Bulge Factor 2 = 15
(Set in Edit Objects)



Bulge Factor 1 = 8; Bulge Factor 2 = 10
(Set in Edit Objects)

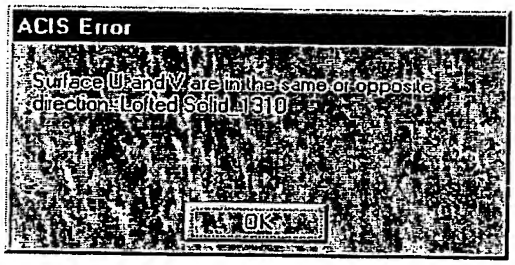
This tool does not support lofting if one of the solids is a sphere. If a sphere is selected no loft is created.

Using the Lofted Solid Tool

1. Select the Lofted Solid tool. The Message Line reads: *Lofted Solid: Pick two faces for lofted solid.*

2. Enter a value in the Bulge data field.

   Be careful not to apply a bulge so large that it results in a self-intersecting body. If it does you will receive the following error message.

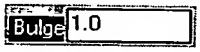


   Also be aware that bulge factors that are too small with respect to the solids may prevent later feature operations such as shelling and blending.

3. Select a solid face.
4. Select a face on the other solid.

   One loft solid is created from the two solids using the shared faces for tangent conditions.

The Status Line contains the Bulge data field.



Geometric Characteristics

A lofted solid is created by selecting a face on two solids. It is made up of the following characteristics according to the Edit Objects dialog box: Bulge 1 and Bulge 2. Bulge 1 applies to the bulge between the first face selected and the lofted solid. Bulge 2 applies to the bulge between the second face selected and the lofted solid. To display the Edit Objects dialog box for the lofted solid only, double-click the lofted solid item in the Design Explorer.

Rib Solid Tool

The rib tool extrudes and thickens a closed or open profile into a base solid. The profile is terminated up to the first face it reaches in the direction of the extrude.

Using the Rib Solid Tool

1. Select the Rib Solid tool. The Message Line reads: *Rib Feature: Pick solid for rib.*

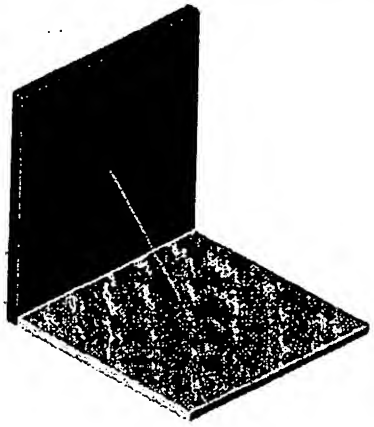
2. Select the solid you want to add the rib to. The Message Line reads: *Rib Feature: Pick rib profile. [Shift = Extend]*

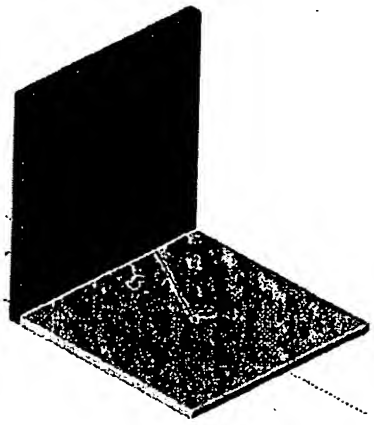
   After you select the rib profile you need to specify the thickness of the rib. This is done in the Status Line entry box pictured here.

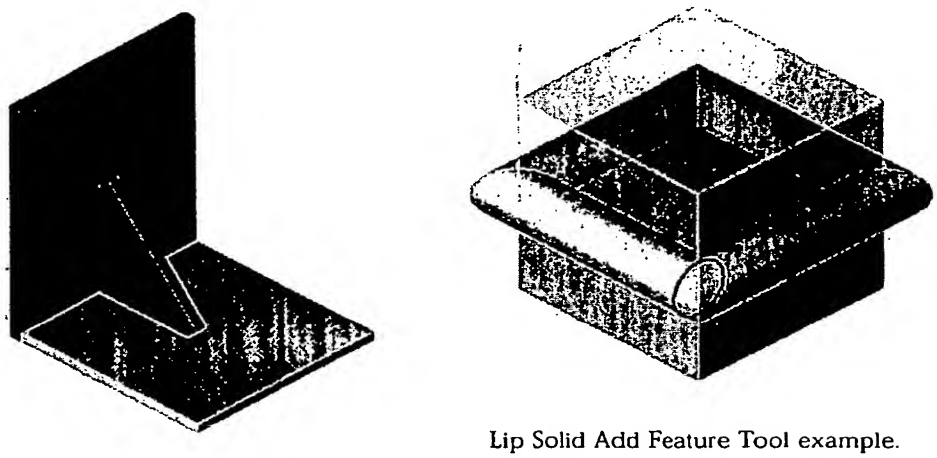
3. Once you select the rib profile The Message Line reads: *Rib Feature: Specify 2 points for rib direction.*

   For the tool to function correctly your direction should run into the solid you are adding the rib feature to.

If you want to edit the Thickness of the rib after you have placed the rib, you must go through the Design Explore and access the Edit Objects box associated with the Rib. The examples below show the tool use progression.







Lip Solid Add Feature Tool example.

Lip Solid Tool

The Lip Solid tool sweeps a profile about a collection of edges and either adds or subtracts the resultant sweep to or from the base solid. The given profile is automatically aligned perpendicular to the selected edge. In addition, a reference point is specified such that the profile is translated from the reference point to the start position along the edge selected. When the base solid is regenerated the profile is always translated from the reference position to that on the edge and then realigned.

Note: If you pick a face for the lip solid tool all edges of the selected face are used for the lip tool.

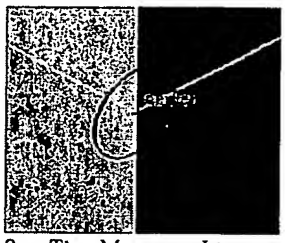
The Lip Solid tool has two options in the Message Line, Add Feature and Remove Feature.

Using the Lip Solid Add FeatureTool

1. Select the Lip Solid tool. The Message Line reads: *Lip Feature Add: Pick curves for lip profile. [Shift = Extend]*

2. Select the curves for the lip profile. The Message Line reads: *Lip Feature: Specify profile origin.*

   The center of the circle is selected to be the origin in this example.

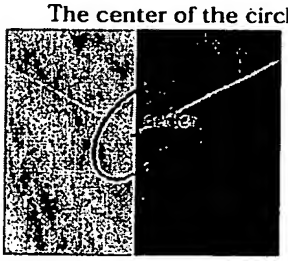


3. The Message Line reads: *Lip Feature Add: Pick edges for profile to sweep. [Shift = Extend]*
4. Pick the edges of the solid you want to add the lip feature to. In the example above, the edges of the green block were chosen.

Using the Lip Solid Remove Feature Tool.

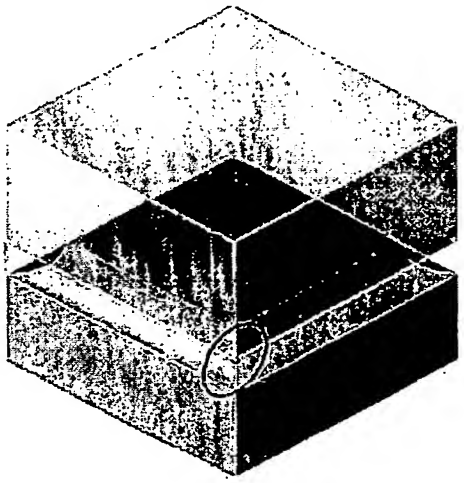
1. Select the Lip Solid tool. The Message Line reads: *Lip Feature Remove: Pick curves for lip profile. [Shift = Extend]*
2. Select the curves for the lip profile. The Message Line reads: *Lip Feature: Specify profile origin.*

   The center of the circle is selected to be the origin in this example.



3. The Message Line reads: *Lip Feature Remove: Pick edges for profile to sweep. [Shift = Extend]*

4. Pick the edges of the solid you want the lip feature profile to be removed from. In the example below, the edges of the green block were chosen.

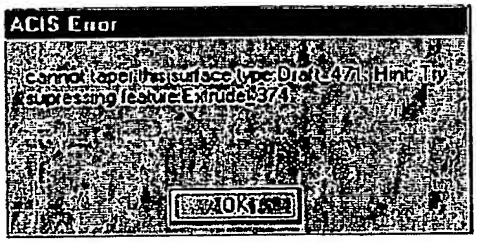


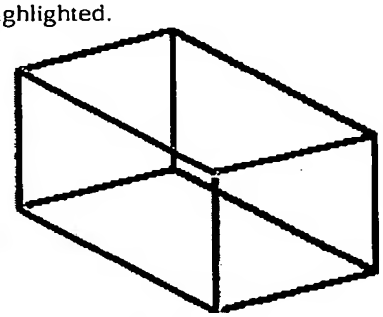
Note: You cannot use the remove feature on both block edges at the same time.

Local Face Modeling Tools

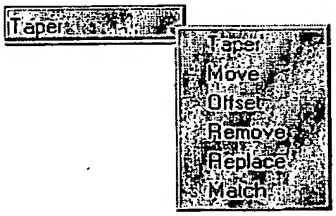
The **Local Face Modeling** tool allows you to perform operations on faces independently of how the part was created. This means you can bring in parts from other systems and still perform edits to the geometry.

This tool also includes an error detection routine. If your model contains a problem that prevents the completion of the local face modeling operation, an ACIS error dialog box appears with the problem location highlighted.





The Local Face Modeling tool offers six modeling options, as listed in the Message Line: *Taper, Move, Offset, Remove, Replace* and *Match*.



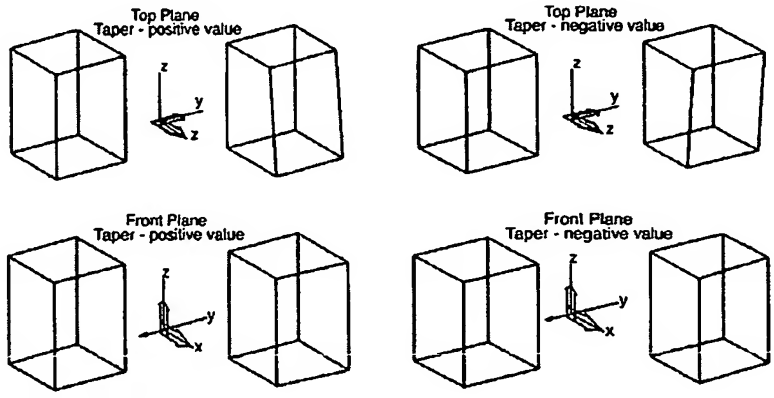
The Draft Face Tool

Choosing the **Draft Face** tool applies a draft to a group of selected faces. Positive tapers (draft) add material. Negative tapers (draft) remove material. The taper direction is relative to the pull-direction defined by the current work plane and a neutral point.

Rules

- The work plane cannot be parallel to the face or faces that you taper. The work plane defines the pull-direction or direction of draw, the direction the part would eject from a mold.
- The neutral point is the position on the face or edge that remains fixed while the face tapers.
- The taper is the angle that the face rotates relative to the neutral point. The value can be negative or positive and the rotation occurs relative to the work plane. Typically positive values add material and negative values remove material. The

graphics here illustrate the relationship between that work plane and the taper angles.

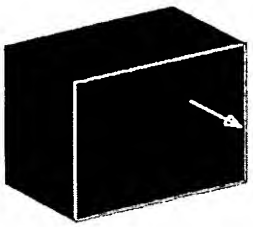


Using the Draft Tool

1. Select the **Draft Solid** tool. The Message Line reads: *Draft Solid: Pick reference face or edge for draft (click nothing for work plane).*

2. Select the reference face, edge or work plane. This tells your Designer Elements program the pull direction of the normal for the taper.

   When you select the reference an arrow displays indicating the normal direction. An example is shown here.



   The Message Line now reads, *Draft Solid: Pick faces of a solid to draft [Shift=Extend].*

Tech Note:

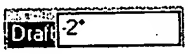
Make sure the face you choose is not normal to the z-axis. If it is, nothing will happen. Change the work plane and try again.

3. Select a face or hold down the SHIFT key to select more than one face. The Message Line now reads, *Draft Solid: Specify taper neutral position.*
4. Click a point on the screen to serve as the neutral position. The neutral position will remain constant.

   Your Designer Elements program applies the taper or draft relative to a pull direction defined by the normal.

While the solid is selected you can change the Draft angle in the Status Line. Press ENTER (Windows) or RETURN (Macintosh) and the taper updates.

The Status Line contains the Draft angle.



Modifying Drafted Solids

You can modify the draft angle of your solid by choosing the draft in the Design Explorer for the solid and displaying the Edit Objects dialog box. If you have more than one face drafted, you can specify a different draft for each one. Click Apply to accept the changes.

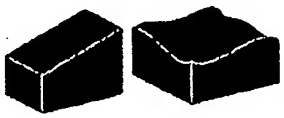
Geometric Characteristics

A draft solid is created by choosing a solid, specifying a draft, the face for the draft and the neutral point position. It is made up of these characteristics according to Edit Objects: Draft angle and the solid face list with the drafts. This list is only available if you select the operation in the Design Explorer first and then display the Edit Object dialog box. See Chapter 24 for more information on the Design Explorer.

The Match Face Tool

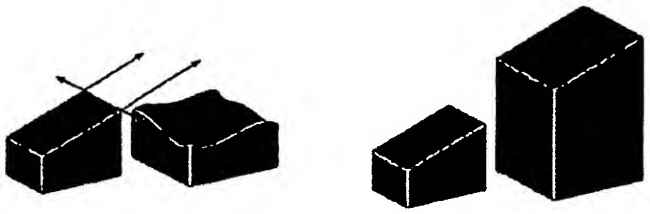
Choosing the **Match Face** tool matches the selected surface to a referenced surface. This tool only works if the referenced surface is analytical (planar or circular). A nurb surface is not an analytical surface.

Planar Example

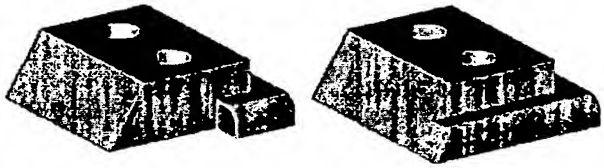


This is an example of matching the top face of the right model to the top face of the left model, which acts as a reference. Imagine the top planar face of the left solid extended beyond its bounds, as shown in the left graphic here. When the operation is complete the nurb face of the right solid is replaced with the planar face, as shown in the

right graphic. The solid is extended to the imaginary intersection with the top planar face of the left solid.



Here's another example of the match face option. The left graphic shows a select face on a solid object. The right graphic shows the face matched to the sloped face.



Elliptical Example

This left graphic below shows an example of matching a side face of the rectangular solid to the elliptical solid, which acts as a reference. When the operation is complete the solid is extended to the elliptical solid and the planar face becomes an elliptical face, shown in the right graphic.

Using the Match Face Tool

1. Select the **Match Face** tool.The Message Line reads: *Match Face: Pick face to*

modify.

2. Select the face of the solid you want to modify or match to another face. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   The Message line now reads; *Match Face: Pick face to match to.*

3. Select the face of the solid that you want the selected face to match.

   The face is matched.

The Status Line contains no entries.

Geometric Characteristics

A matched face has no specific characteristics. To modify the Display and Attribute characteristics of the matched face, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

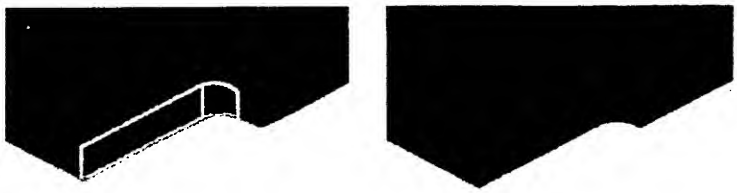
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

See Chapter 24 for more information on the Design Explorer.

The Move Face Tool

Choosing Move Face tool moves the selected group of faces by a vector defined with the Drafting Assistant.

If moving more than one face the selected faces cannot be perpendicular to others. This tool is valuable for moving a flat face and fillet or a face and a hole.The graphics here show just such an operation. The left graphic shows the original geometry with the selected faces and the right graphic shows the geometry with two faces moved.



Using the Move Face Tool

1. Select the Move Face tool.The Message Line reads: *Move Face: Pick faces of a solid to move [Shift=Extend].*

2. Select the face(s) of the solid you want to move. To select more than one face hold down the SHIFT key before selecting the first face.

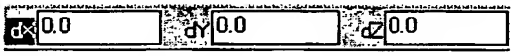
   The Message Line now reads, *Move Face: Pick two points for move.* The points define a vector with DX, DY, DZ values. You can either click two points or enter the desired values in the Status Line.

3. Select the two points to specify dX, dY, dZ values for the move. The order you select your points determines whether the value is positive or negative.

   The selected face moves to the new location.

While the face is still selected you can enter new values in the dX, dY, or dZ data fields in the Status Line. Press ENTER (Windows) or RETURN (Macintosh) to update the move.

The Status Line contains the dX, dY and dZ values of the move.



Geometric Characteristics

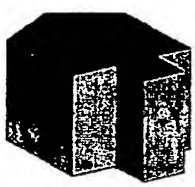
A move face has no specific characteristics. To modify the Display and Attribute characteristics of the move face, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

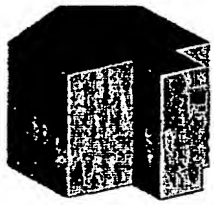
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

See Chapter 24 for more information on the Design Explorer.

The Offset Face Tool

Choosing the Offset Face tool offsets the selected faces. Adjacent faces and edges are adjusted as necessary for the offset. The left graphic below shows the original object. The right graphic shows a positive offset of the left and top face.





Tech Note:

When using the *Move Face* option, you will receive an error if you try to move the whole face off a solid.

Using the Offset Face Tool

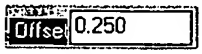
1. Select the Offset Face Modeling tool.

2. The Message Line reads: *Offset Face: Pick faces of a solid to offset [Shift=Extend].*
3. In the Status Line enter a value in the data field. A positive value enlarges the object and a negative value reduces the object.
4. Select the face of the solid you want to offset. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   The selected face moves the offset specified.

While the object is still selected you can enter new values in the data field. Press ENTER (Windows) or RETURN (Macintosh) to update the offset.

The Status Line contains the Offset value.



Geometric Characteristics

An offset face is defined by its offset value according to the Edit Objects dialog box. To modify the offset or the Display and Attribute characteristics of the offset face, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On. See Chapter 24 for more information on the Design Explorer.

The Remove Face Tool

Choosing the Remove Face tool removes a face from a solid, extends the adjacent faces and relimits the solids as necessary. This is useful for removing holes and fillets from geometry imported without a history tree. The left graphic shows the original object. The right graphic shows the object with the holes and a fillet removed.

Using the Remove Face Tool

1. Select the Remove Face tool.
2. The Message Line reads: *Remove Face: Pick faces of a solid to remove [Shift=Extend].*
3. Select the face of the solid you want to remove. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   The selected face is removed.

There are no Status Line entries.

Warning: A face can only be removed from a solid as long as the integrity of a solid is retained after the face removal. You cannot remove a face of a cube if the

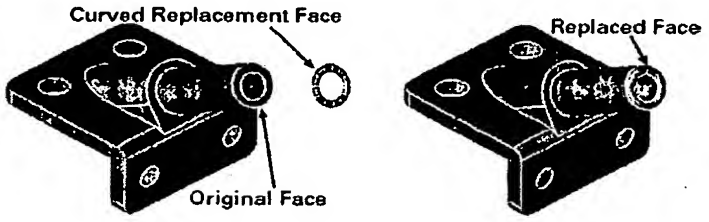
cube has a blend, the blend can be removed. If you attempt to remove a face that will challenge the solid's integrity, the following error message appears.

Geometric Characteristics

A removed face has no specific characteristics.

The Replace Face Tool

Choosing the Replace Face tool replaces the selected face with the new face specified. The new face must have the same boundaries or edges as the original face.



Using the Replace Face Tool

1. Select the Replace Face tool.

2. The Message Line reads: *Replace Face: Pick face to replace.*
3. Select the face of the solid you want to replace. To select more than one face be sure to hold down the SHIFT key before selecting the first face.

   The Message line now reads, *Replace Face: Pick new surface.*

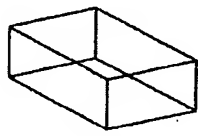
4. Select the replacement face.

   The original face is replaced.

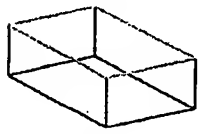
The Status Line contains no entries.

Replace Face Example

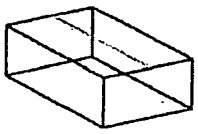
You have a block (below) and you want to replace the top face with a warped surface.



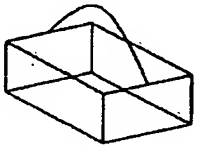
1. Select the **Rectangle** tool and trace over the top face of the block.



2. Select the **Through Point B-Spline** tool and create a three point spline from the midpoint of the of one edge to the midpoint of the opposite edge.

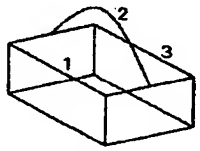


3. Using the **Selection** tool, drag select the center control point of the spline and drag it up along the z axis some distance.

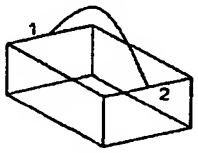


4. Select the **Net Surface** tool.

5. Select the M curves consisting of the left edge, spline and right edge curves, (curves 1, 2 and 3 in the graphic).

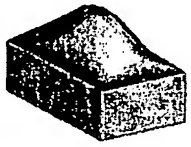


6. Select the N curves consisting of the top and bottom curves (curves 1 and 2 in the graphic) two complete the net surface.
7. Select the *Replace Face* tool.



8. Select the top face of the block to replace.
9. Select the net surface you just created to replace the face.

   The top face of the solids is replaced.
10. Now render the block.



Geometric Characteristics

An replace face has no specific characteristics. To modify the Display and Attribute characteristics, select the face using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name.

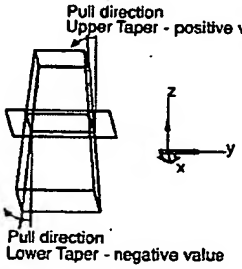
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On. See Chapter 24 for more information on the Design Explorer.

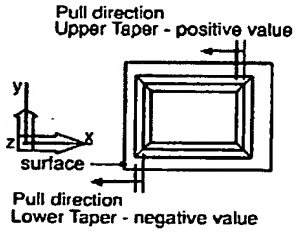
Parting Line Solid Tool

The Parting Line tool automatically applies draft angles to a collection of upper and lower surfaces. These faces are determined by the location of the parting line curve. This Designer Elements program uses curves as the parting entity so you must create the curves before using the tool.

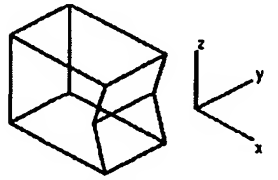
Rules

- The work plane cannot be parallel to the face or faces that you taper. The work plane defines the pull-direction or direction of draw, the direction the part ejects from a mold.

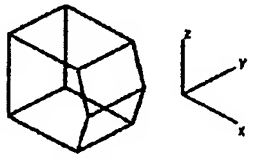




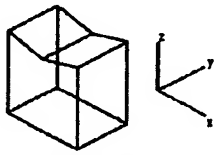
- The taper is an angle that the face rotates relative to the neutral point. The value can be negative or positive and the rotation occurs relative to the work plane. Typically, positive values add material and negative values remove material. The graphics here illustrate the relationship between that work plane and the taper angles.



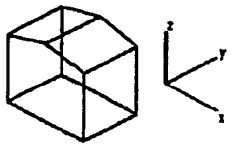
Top Plane
Upper Taper - positive value
Lower Taper - positive value



Top Plane
Upper Taper - negative angle
Lower Taper - negative angle



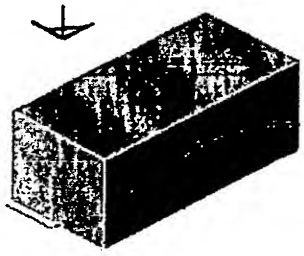
Side Plane
Upper Taper - positive value
Lower Taper - positive value

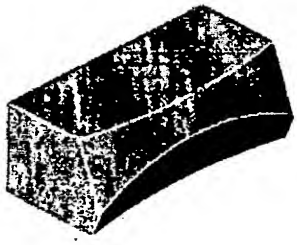


Side Plane
Upper Taper - negative angle
Lower Taper - negative angle

- If you have difficulty placing a draft on a solid try changing the work plane and attempting the draft again.

The left graphic shows an object with a parting line curve. The right graphic shows the object after using the tool.





Using the Parting Line Solid Tool

1. Select the Parting Line tool. The Message Line reads: *Parting Line Solid: Pick solid for parting line.*

2. Select the solid that you want to part. The Message Line reads: *Parting Line Solid: Pick curves for parting line.*
3. Select the parting line curves. To select more than one curve be sure to hold down the SHIFT key before selecting the first curve.

While the solid is still selected you can change the upper and lower taper values. Press ENTER (Windows) or RETURN (Macintosh) and the part updates.

The Status Line contains the Upper Taper and the Lower Taper angle for the solid.

Modifying Parting Line Solids

You can modify the taper angles of your solid by choosing the taper in the Design Explorer for the solid and displaying the Edit Objects dialog box. You can modify the taper angle for the Upper and Lower Tapers. Click Apply to accept the changes.

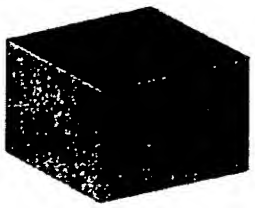
Geometric Characteristics

A parting line solid is created by choosing a solid, intersecting curve and the upper and lower taper. It is made up of these characteristics according to Edit Objects: Draft angle and the solid face list with the tapers. This list is only available if you select the operation in the Design Explorer first and then display the Edit Object dialog box. See Chapter 24 for more information on the Design Explorer.

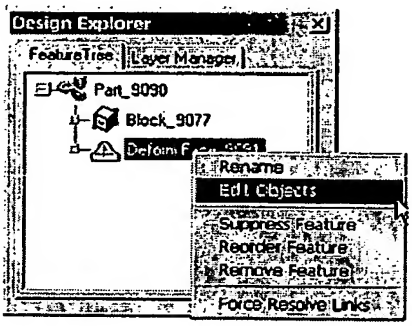
Deform Face Tool

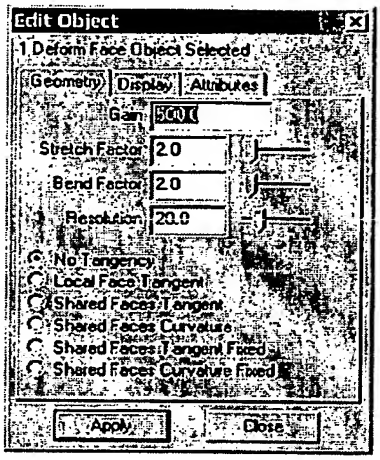
The Deform Face tool allows you to deform surfaces and solid faces. When you select the Deform Face tool, a subpalette appears in the Message Line containing three tools, Deform with Gain, Deform to Point and Deform to Curve.

All deform face examples will use the solid block here to illustrate each Deform Face tool.



After you have used the Deform Face tool and continue building you model and then you decide you want to edit the deformed face you must access the Edit Objects dialog box through the Design Explorer.





The Deform Face Edit Objects box contains the following options:

Gain

The gain value is similar to a constraint pressure applied to the surface. Use positive gain to inflate the surface and negative to deflate. Depending on the stiffness and resolution of the surface practical gains can range anywhere from 0 to 1000000.

Stretch Factor

The stretch factor controls the deformable surface's resistance to stretching. A surface with a large stretch value is said to be stiff. Deformable surfaces with large stretch values act like soap bubbles seeking to always minimize their area. This results in flatter looking surfaces that allow regions of rapid bending.

Bend Factor — The bend factor controls the deformable surface's resistance to bending. Deformable models with large bend values act like elastic beams attempting to distribute regions of bending over large areas and typically generate very fair shapes.

Resolution — The resolution slider controls the precision of the resultant deformed shape by inserting additional control points to the surface. Higher resolution values will show more detail for the given deformation values. Lower resolution values calculate faster but with less detail. When using the tangent and curvature options, start with a resolution factor of 80 for best results.

The following options apply to only to pressure deformations, the first tool icon. It does not apply to deformations to a point or curve. Be sure to use higher resolutions when using the below options to insure tangency and curvature precision with the results.

No Tangency — The No Tangency option deforms the face and allows the shape at the edges to deform.

Local Face Tangent — The Local Face Tangent option deforms the shape but preserves the existing tangencies of the face.

Shared Faces Tangent — Shared Face Tangent will modify all faces that share edges with the face being modified. The modification will impose tangencies at all shared edges.

Shared Faces Curvature — The Shared Face Curvature option will modify all faces that share edges with the face being modified. The modification will impose curvature continuity at all shared edges.

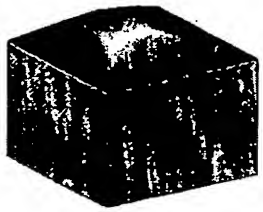
Shared Faces Tangent Fixed — The Shared Face Tangent Fixed option will only modify the selected face to be tangent to all faces that share an edge with the selected face.

Shared Faces Curvature Fixed — The Shared Face Curvature Fixed option will only modify the selected face to be curvature continu-

ous to all faces that share an edge with the selected face.

Deform Face with Gain Tool

This tool applies a pressure gain (negative or positive) to the selected surface or face. The gain displays in the Status Line. Large gains distort the object while small gains barely move the object. The pressure applies to the entire surface or face. You cannot specify a location for the deformation. Use the **Deform Face to Point** tool for that ability. The graphic shows the top face deformed using this tool.



Using the Deform Face with Gain Tool

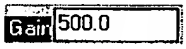
1. Select the **Deform Face** tool.
2. Select the **Deform Face with Gain** tool in the Message Line. The Message Line reads: *Deform Face: Pick face to deform with pressure.*
3. Type the desired pressure in the Gain data field of the Status Line.

   The number can be a positive or negative value.
4. Select the face or surface that you want to deform.

   The object is deformed. Render your object using any render option other than wireframe to view the deformation.

While the object is still selected you can enter a new value in the Gain data field and press ENTER (Windows) or RETURN (Macintosh) to change the gain.

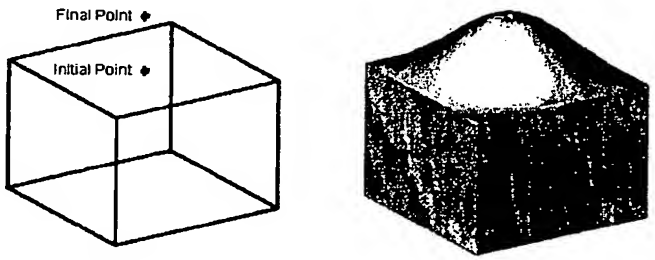
The Status Line contains the Gain value for the deformation.



Deform Face to Point Tool

This tool applies a pressure gain (negative or positive) to the selected surface or face passing through a specific point in space. You choose a point on the object to begin the deformation and the point's final location. The left graphic shows the ini-

tial point on a solid face and its final location. The right graphic shows the rendered object and the point.



The pressure gain displays in the Status Line.

Using the Deform Face to Point Tool

If you want the deformation to pass through a point not already in your drawing, create the point before using this tool.

1. Select the **Deform Face** tool.
2. Select the **Deform Face to Point** tool in the Message Line. The Message Line reads: *Deform Face: Pick face to deform.*
3. Type the desired pressure in the Gain data field of the Status Line.
4. Select the surface or face that you want to deform.

   The Message Line reads: *Deform Face: Pick location to deform from.*
5. Click a location on the selected surface/face to set the point from which the face deforms.

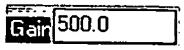
   The Message Line reads: *Deform Face: Pick location to deform to.*
6. Click the final location for the face point where the deformation begins.

   The object deforms. Render your solid using any render option other than wire-frame to view the deformation.

While the surface or face is still selected you can enter a new value in the Gain data field and press ENTER (Windows) or RETURN (Macintosh) to change the gain.

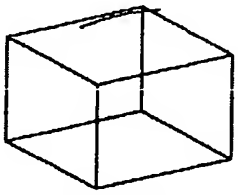
You can move the point later by dragging it to a new location or using the Edit Objects dialog box when the point is selected. The deformed object automatically updates.

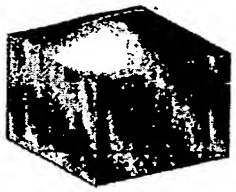
The Status Line contains the Gain value for the deformation.



Deform Face to Curve Tool

This tool applies a pressure gain (negative or positive) to the selected surface or face attached to a specified curve. You create the curve before using this tool. The left graphic shows the object with the curve. The right graphic shows the rendered image and the curve.





The pressure gain displays in the Status Line. A high enough gain forces the surface or face to pass through the curve. The curve is associative to the deformation. Moving the curve changes the associated deformation.

Using the Deform Face to Curve Tool

If you want the deformation to use a curve not already in your drawing, create the curve before using this tool.

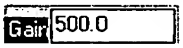
1. Select the **Deform Face** tool.
2. Select the **Deform Face to Curve** tool in the Message Line. The Message Line reads: *Deform Face: Pick face to deform to curve.*
3. Type the desired pressure in the Gain data field of the Status Line.
4. Select the surface or face that you want to deform.

   The Message Line reads: *Deform Face: Pick curve constraint.*
5. Select the curve.

   The object is deformed. Render your object using any render option other than wireframe to view the deformation.

While the object is still selected you can enter a new value in the Gain data field and press ENTER (Windows) or RETURN (Macintosh) to change the gain.

The Status Line contains the Gain value for the deformation.



Geometric Characteristics

A deformable surface or face is created by selecting object and using pressure, a point or curve. This deformed surface or face is made up of the following characteristics according to the Edit Objects dialog box: Gain, Stretch Factor, Bend Factor and Resolution. The dialog box also contains the Keep Boundary Tangents option.

Gain — This field displays the pressure applied to the surface or face. You can enter a different gain and click Apply.

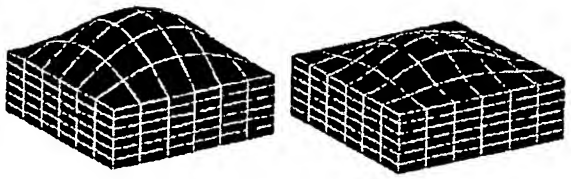
Stretch Factor — This field sets the resistance of the surface or face to the stretching. Higher values result in a flatter surface. Lower values result in a smoother surface. Enter a value yourself or use the slide to specify a factor.

Bend Factor — This field sets the surface or face resistance to bending. Higher values prevent sharp radical changes to the surface. Enter a value yourself or use the slide to specify a factor.

Resolution — This field sets the number of internal grids used to calculate the deformation. Higher values increase the influence of the bend and stretch factors to pressure gains. Lower values decrease the influence, providing quick results but few discriminating features to your surface. Enter a value or use the slide to specify a resolution.

Keep Boundary Tangents — This option determines whether the edges of the surface or face remain tangent. When selected, the edges are tangent. This left graphic here shows the top deformed solid face without tangent edges.

The right graphics shows the deformed face with tangent edges.



To modify the Display and Attribute characteristics of the deformed surface or face select it using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the item's name.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

See Chapter 24 for more information on the Design Explorer.

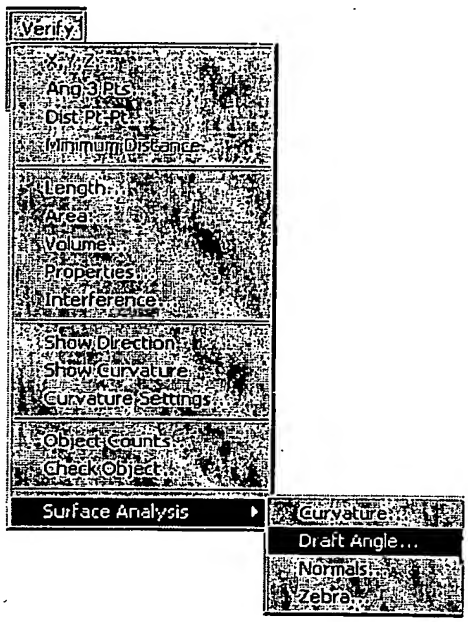
Draft Evaluation Feature

This Designer Elements program gives you the ability to evaluate the drafts of an object for molding purposes. This feature is available through the Verify menu and the advanced rendering capabilities.

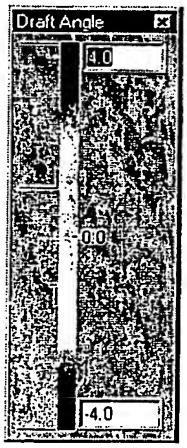
Draft Evaluation through Verify Menu

The *Draft Angle* in the Verify menu allows you to quickly evaluate the draft angles of a model.

All analysis colors are preset with this commands. If you would like to define your own settings, use the draft evaluation shader through the Advanced Rendering feature. See "Draft Evaluation through Advanced Rendering" on page 40.



This command evaluates the drafts of an object. When you select this command, the Draft Angle dialog box appears with the draft angle analysis, containing the histogram with the analysis, the color spectrum and histogram data fields.



The dialog box contains the following elements:

Histogram

Located to the left of the color spectrum in the dialog box, these horizontal bars represent the frequency of a draft angle using the color spectrum. The length of the bar represents the frequency.

This Designer Elements program calculates this histogram

and displays it so that the entire graph fits into the dialog box area.

Color spectrum bar

The section displays the color septum used to create the histogram.

Histogram data fields

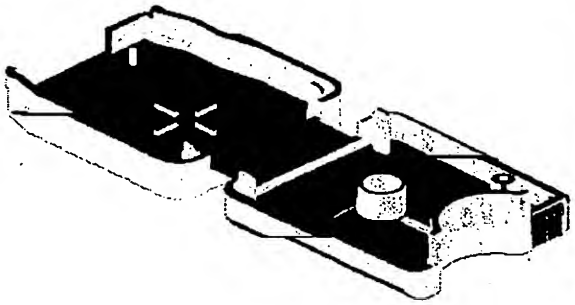
The data fields display the maximum and minimum values used to calculate the histogram. When you initially select the command, this Designer Elements program scans the surface(s) and sets the min and max fields for the draft angle analysis.

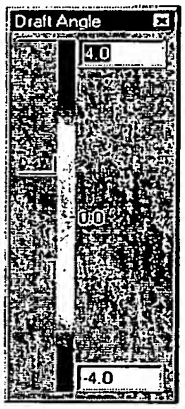
If you enter different values in the fields such that a large number of values fall outside of the specified range a red line appears at the end(s) where the values fall. All values are still calculated, even though they are not displayed due to the specified range.

If you make changes to the values, this Designer Elements program waits for two seconds of non-action before recalculating the histogram, giving you time to change the values before the image is rendered again.

You can return the range initially displayed by clicking on the selected option again. (The option does not deselect when you do this.) Windows users can also reset the maximum and minimum values by right mouse clicking near the respective end of the spectrum.

The graphic here is an example of a draft angle analysis.

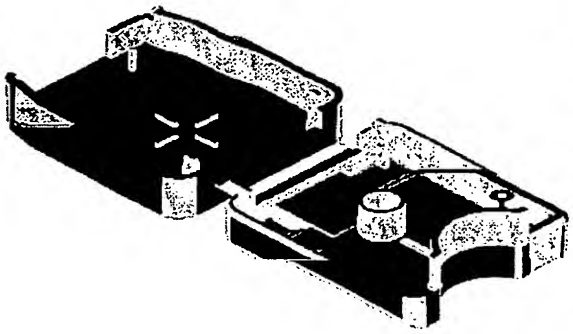


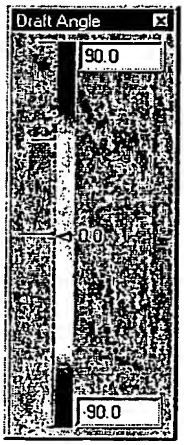


Using the Draft Angle Command

1. Select the object.
2. Choose ***Verify>Surface Analysis>Draft Angle***.

   The dialog box displays with the analysis. Your geometry also displays the analysis.
3. To display a certain analysis/color area, place the pointer at the location over the color spectrum and click the mouse.



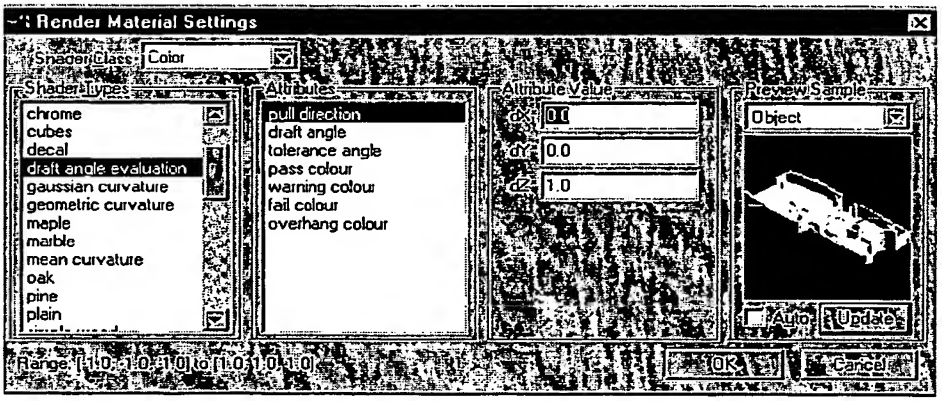


Notice that a triangular indicator appears at the selected location and the related color highlights in the histogram. The same color highlights in black on your object.

4. To remove the triangular indicator click in the dialog box outside of the color spectrum bar.
5. Change the histogram values in the data fields and the histogram and analysis automatically recalculates.

Draft Evaluation through Advanced Rendering

Using the Advanced Rendering feature you can specify your own draft angle settings. When you display the Render Material Settings dialog box and choose the draft angle evaluation type, the dialog box appears.



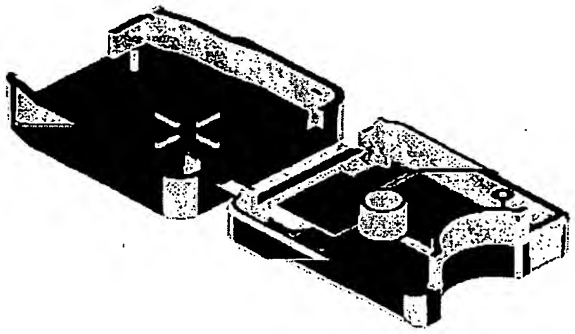
The draft evaluation type contains the following attributes:

pull direction — This specifies the direction the object is pulled from the mold. Enter the values for the pull direction or drag in the drawing area to have the values automatically entered. The asterisk next to the dX, dY and dZ names indicate this ability.

draft angle — This specifies the draft angle required to pull the object out. This angle is usually 1°. Entering a 0 results in the fail color zone disappearing when the scene is rendered.

tolerance angle	This specifies the degree tolerance added to the draft angle that still allows the object to be pulled from a mold but with difficulty. Entering a 0 results in the warning color zone disappearing.
pass color	This specifies the color of the object for those areas where the draft angles are satisfactory. The surface normals fall between 90°+ **draft angle** + **tolerance angle** and 180° with the pull direction.
warning color	This specifies the color that appears on the object when the surface normals fall between 90°+ **draft angle** and 90°+ **draft angle** + **tolerance angle**.
fail color	This specifies the color of the object in those areas where the object cannot be pulled from the mold. The surface normals fall between 90° and 90°+ **draft angle**.
overhang color	This specifies the color that appears for any part of the mold overhanging the object. An overhang on the mold prevents the object from being pulled from the mold.

In the graphic here the pass color areas are medium gray, the overhang color areas are black and the warning color areas are light gray.



Using the Draft Evaluation Feature

1. Double click on the object you want to evaluate to display the Edit Objects dialog box.

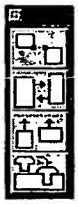
A material must already have been placed on the object.

2. Click the Advanced button to display the Render Material Settings dialog box.
3. Select draft angle evaluation from the Shader Types list.
4. Specify the settings for the attributes.
5. Click OK to close the dialog box and save the settings.
6. Render your object using one of the photorealistic commands.

   The object displays using your specified draft evaluation colors and values.

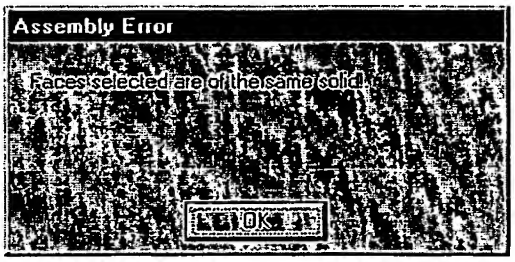
Assembly Modeling Tools

This Designer Elements program provides you with tools for doing assembly modeling. These tools are located in their own tool palette. Choose ***Window>Assembly*** to display the tool palette.



The Assembly Modeling tools include: Assembly Connect, Assembly Mate, Assembly Align and Assembly Insert.

Each tool performs a particular operation involving two solids. The following error appears if you attempt to perform an operation on a single solid.



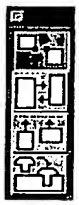
Assembly Connect Tool

The Assembly Connect tool to connects one solid to another at a specified connection point. The resulting part is associative. When you move one solid the other

moves as well to maintain the connection. The left graphic shows two solid objects. The right graphic shows the solids after connecting them.

This tool supports connection points along object edges including start, end and middle and on locations recognized by the Drafting Assistant (*center, midpoint, endpoint,* etc.). You cannot select more than one solid to connect at a time.

Using the Assembly Connect Tool



1. Select the Assembly Connect tool. The Message Line reads: *Assembly Connect: Pick solid to move.*
2. Select the solid.

   The Message Line now reads, *Assembly Connect: Pick the connection point location on selected solid to move.*
3. Using the Drafting Assistant select the connection point on the selected solid model.

   The Message Line reads: *Assembly Connect: Pick solid to move to.*
4. Select the solid to which you are moving the first solid.

   The Message Line reads: *Assembly Connect: Pick new location on the selected solid for first solid selected.*
5. Using the Drafting Assistant select the connection point on the selected solid to which you want to move the first solid selected.

   The two solids are connected.

The Status Line contains no entries.

Geometric Characteristics

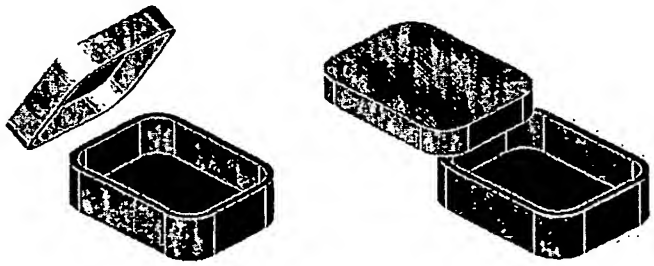
A connected assembly is made up of two solids. It does not include an characteristics that are unique. To modify Display and Attribute characteristics of an assembly, select the object using the Design Explorer and choose ***Window>Edit Objects*** or

double-click on the face name. See Chapter 24 for more information on the Design Explorer.

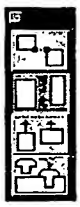
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

Assembly Mate Tool

The Assembly Mate tool mates planar faces. You can specify an offset distance between mated faces. The normals of the mated faces point in opposite directions. This tool only ensures that the selected faces are co-planar. If you want to align the faces, use the Assembly Align tool. The left graphic shows the two solids. The right graphic shows the solids after mating them.



Using the Assembly Mate Tool



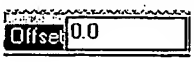
1. Select the Assembly Mate tool. The Message Line reads: *Assembly Mate: Pick solid face to mate.*
2. Select the solid face you want to mate.

   The Message Line now reads, *Pick solid face to mate with.*
3. Select the solid face you want to mate with.

   The two solids are now mated.
4. If you want the first solid offset a certain distance from the second, enter a value in the Status Line data field and press ENTER (Windows) or RETURN (Macintosh).

The objects are offset the specified distance but are still co-planar.

The Status Line contains the Offset data field.



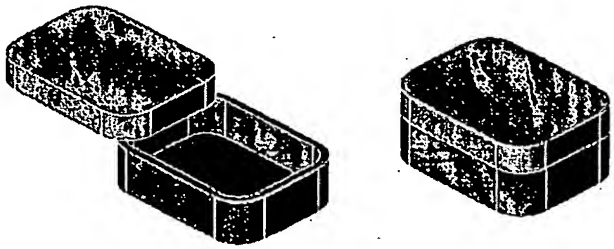
Geometric Characteristics

A mate assembly is made up of two mated solid objects and includes an Offset value according to the Edit Object dialog box. To modify the characteristics of a mate assembly, select the assembly using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name. See Chapter 24 for more information on the Design Explorer.

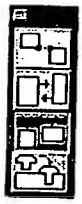
Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

Assembly Align Tool

The Assembly Align tool aligns a face of two solid objects and supports cylindrical, spherical and planar faces. This tool reorients the parts so that the faces are aligned in the same direction. The left graphic shows the two solid. The right graphic shows the solids after aligning them.



Using the Assembly Align Tool



1. Select the Assembly Align tool. The Message Line reads: *Assembly Align: Pick solid face to align.*
2. Select the solid face you want to align.

   The Message Line now reads, *Assembly Align: Pick solid face to align with.*
3. Select the solid face to align with.

   The two solids are now aligned.

The Status Line contains no entries.

Geometric Characteristics

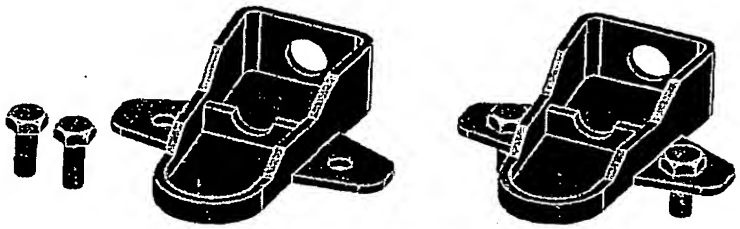
An aligned object is made up of two aligned solids and includes an Offset value according to the Edit Object dialog box. To modify the characteristics of an aligned object, select the object using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name. See Chapter 24 for more information on the Design Explorer.

Display contains settings for Iso Lines and Silhouette. You can display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

Assembly Insert Tool

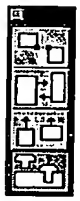
The Assembly Insert tool aligns and mates the faces of two solid objects. The align faces must be cylindrical and the mate faces must be planar. This tool is very useful

for inserting bolts into holes. The left graphic shows two bolts and another solid. The right graphic shows the bolts inserted into the solid.



In the example, the bolt cylinder is aligned with the circular hole and its planar face mated with the planar face of the other solid though which the hole passes.

Using the Assembly Insert Tool



1. Select the **Assembly Insert** tool. The Message Line reads: *Assembly Insert: Pick solid face to align.*
2. Select the solid face you want to align.

   The Message Line now reads, *Assembly Insert: Pick solid face to align with.*
3. Select the solid face to align with.

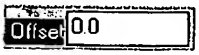
   The Message Line reads: *Assembly Insert: Pick solid face to mate.*
4. Select the solid face you want to mate.

   The Message Line now reads, *Assembly Insert: Pick solid face to mate with.*
5. Select the solid face you want to mate with

   The first solid is now inserted into the second solid.
6. If you want the first solid offset a certain distance from the second, enter a value in the Status Line data field and press ENTER (Windows) or RETURN (Macintosh).

   The objects are offset the specified distance but are still co-planar.

The Status Line contains the Offset data field.



Geometric Characteristics

An inserted object is composed of two aligned and mated solids and includes an Offset value according to the Edit Object dialog box. To modify the characteristics of an inserted object, select the object using the Design Explorer and choose ***Window>Edit Objects*** or double-click on the face name. See Chapter 24 for more information on the Design Explorer.

Display contains settings for Iso Lines and Silhouette. Display Iso Lines by entering the value in the U and V data fields and clicking Apply. A grid of U and V lines will display on the object. You have three Silhouette settings, Smart, Off and On.

Editing Commands

This Designer Elements program provides a variety of editing commands to enable you to edit your geometry quickly an easily. Some commands deal with the placement or your geometry like *Cut* and *Align*. Some commands change the appearance of your object like *Change Resolution*. Still others change your geometry like *Change Object Type* and *Edit.Objects*.

The chapter includes these sections:

- Basic Editing Commands
- Advanced Editing Commands
- Object Commands
- Verify Menu

Basic Editing Commands

These commands provide some standard editing features like: *Cut*, *Paste*, *Undo* and *Redo*.

Cut - CTRL+X (Windows); z +X (Macintosh)

This command in the Edit menu removes the selected object(s) and places them on the Clipboard. Each selection you cut or copy to the Clipboard replaces the previous Clipboard contents.

Cutting Objects

1. Select the object to be cut.
2. Choose ***Edit>Cut***.

Once you cut a selection you can paste it. You can use *Cut* and *Paste* to move geometry or text around the document or from one document to another. You can even paste the cut selection onto a document in a different application.

Moving Geometry with the Cut Command

1. Select the object(s) you want to move.
2. Choose ***Edit>Cut***.
3. Display the location where you want the selection to appear in the drawing area, scrolling if necessary.
4. Choose ***Edit>Paste***.

The object appears in the center of the drawing area on the original layer on which it was created or onto the work layer if it's from a different application. The object is selected so you can move it, if you want.

Copy - CTRL+C (Windows); z +C (Macintosh)

Tech Note:

Do not use *Copy* and *Paste* to create an instance of a solid. By doing so you will break the parent/child relationship. **(Cobalt™ and Xenon™ only)** Use the **Move** tool instead. See Chapter 25 for more information.

This command in the Edit menu places a copy of the selection onto the Clipboard without deleting the original selection. You can paste the copy elsewhere in the current document or into a different document. You can even paste the copied selection into a document created with a different application.

Copying Objects

1. Select the object(s) to be copied.
2. Choose ***Edit>Copy***.

   The selection goes on the Clipboard and it remains in the current document.

Copy Pict/Copy Bitmap

By using the *Copy Bitmap* (Windows) or *Copy Pict* (Macintosh) command you can specify a region to copy as a Raster image. You can then paste the image back into the same document, another document or a different application.

Paste - CTRL+V (Windows); z +V (Macintosh)

This command in the Edit menu pastes a copy of the Clipboard contents onto the center of the drawing area. The Clipboard contents are not changed when you use the *Paste* command.

You can also paste the selection into another document or into a document created with another application.

If you want to create an even distribution of geometry, use the **Linear Duplicate** tool or the **Polar Duplicate** tool both are located in the Transformation tools palette.

Tech Note:

You cannot use the *Copy* and *Paste* commands to copy entries in data fields like in the Status Line and dialog boxes. For Windows, press the right mouse button and use the commands in the menu. For Macintosh, use z +C to copy and z +V to paste.

Rules:

1. If the object was cut or copied from a Designer Elements program document, when it is pasted into another Designer Elements program document it goes on the work layer regardless of the layer from which it was cut or copied. A pasted selection retains its original attributes.
2. Objects cut or copied from non-Ashlar-Vellum documents are pasted onto the work layer.

The Clipboard

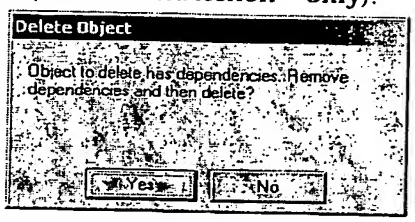
The *Cut* and *Copy* commands place a copy of the selected object on the Clipboard. The Clipboard is a buffer—a temporary storage place that holds the last cut or copied selection. The contents of the Clipboard are objects, not bitmaps or picts (unless you chose *Copy Bitmap/Pict*).

Erasing/Deleting Geometry

This Designer Elements program provides two methods for erasing objects:

- You can select the object(s) then press the BACKSPACE (Windows) or DELETE (Macintosh) key. (You can retrieve what you deleted with the *Undo* command.)
- You can select the object(s) then choose ***Edit>Cut***. (You can retrieve what you cut by using the *Paste* command, as long as you have not cut or copied anything else.)

If you try to delete an object that is parent in a parent/child relationship, this warning message will appear (**Cobalt™ and Xenon™ only**):



When you delete objects in this Designer Elements program the data is still retained in the file until you delete again. If you delete objects before closing a file, perform the operation again to ensure that the data is removed.

Undo - CTRL+Z (Windows); z +Z (Macintosh)

This command in the Edit menu reverses the last action. You can undo an infinite number of times in this Designer Elements program. When you choose *Undo* the last action taken is displayed next to the command.

You can *Undo* actions that create and edit geometry and text but not actions that do not change the contents of the drawing such as resizing the window or quitting.

If you are using a tool which involves a multi-step process, such as creating a 3-Entity Fillet, choosing *Undo* reverts to the beginning of the process. Some activities can be terminated by pressing the ESC key.

Redo

This command in the Edit menu reverses the *Undo* command. You can redo an infinite number of times in this Designer Elements program. When you choose *Redo*, the last action taken is displayed next to the command.

You can *Redo* actions that create and edit geometry and text but not actions that do not change the contents of the drawing such as resizing the window or quitting.

Advanced Editing Commands

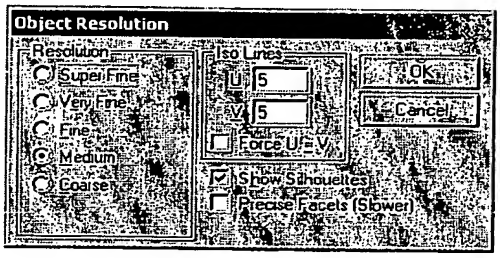
These commands provide more specific editing features/functions involving location, selection and display.

Change Resolution

This command, located in the Edit menu, changes the display resolution of curves, surfaces and solids.

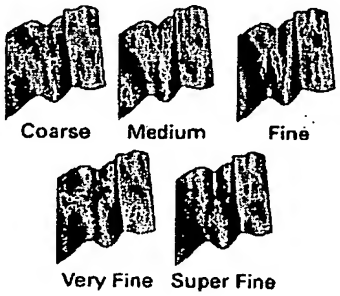
A higher resolution means that this Designer Elements program uses more line segments to display curves on the screen so they look smoother but take longer to draw.

For surfaces and solids a higher resolution also means that this Designer Elements program uses more facets to display the objects. You can control the number of isolines used to display surfaces with this command.



The dialog box contains the following options:

Resolution This section contains radio buttons for five resolution types *Course, Medium, Fine, Very Fine* and *Super Fine.*



Iso Lines This section contains the data fields for *U* and *V* isopram (Iso) lines and the *Force U=V* check box.

The U and V data fields allow you to set the number of iso lines drawn for a surface. Iso lines are constant parameter curves that lie on a surface typically defined in parameter space. The parameter

space coordinate system uses U and V coordinates where U = horizontal and V = vertical. A zero (0) in both fields turns off Iso lines. The appropriate U/V values may enhance the visual appearance of the surface at the expense of drawing speed. The letters, U and V are industry standard space coordinates.

The Force U=V check box automatically sets equal number of U and V lines. Entering a new value in one automatically changes the other isoline value.

Show Silhouettes — This check box controls whether this Designer Elements program draws the silhouette edges of objects. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. A check in the box turns on the silhouette.

OK — Click this button to close the dialog box and save the new settings.

Cancel — Click this button to close the dialog box without saving the changes.

Using the Change Resolution Command

1. Select the object whose resolution you want to change.
2. Choose ***Edit>Change Resolution***. The Object Resolution dialog box appears.
3. Select the desired resolution.
4. Click OK to save the resolution and close the dialog box.

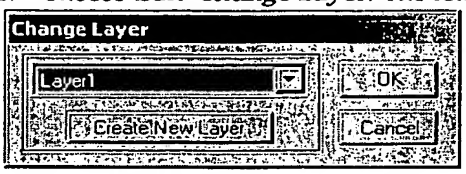
Change Layer

This command in the Edit menu moves selected objects to a different layer and creates new layers.

Using the Change Layer Command

1. Select the object that you want to move to a different layer.

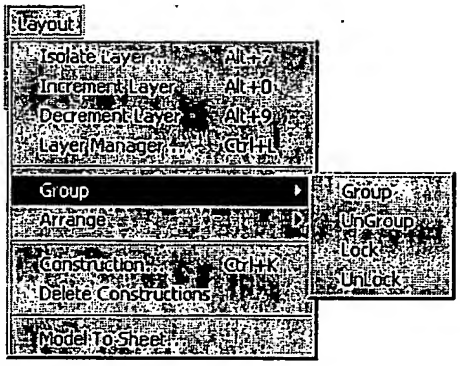
2. Choose ***Edit>Change Layer***. The following Change Layer dialog box appears:



Click on the pull-down menu and select the layer to which you want to move the selected object. If the layer doesn't exist click the Create New Layer button. A new layer appears in the layer field. Enter a new name if you wish.

Click OK to close the dialog box. The selected object is now placed on the selected layer.

Grouping Objects



A group is a collection of objects or short cuts that this Designer Elements program treats as one unit. When you move a group all members of the group move as a single unit. If you delete a group all members are deleted.

Group Command

This command located in the Layout menu includes a submenu with these commands: *Group*, *Ungroup*, *Lock* and *UnLock*.

Using the Group Command

1. Select the objects to be grouped.
2. Select ***Layout>Group>Group***.

Creating temporary groups is useful when moving multiple objects. You can drag a selection fence around several objects to treat them as a single unit while they are selected.

This Designer Elements program supports nested groups. You can create a group and then include it in a second group with other objects or groups.

If you want to add new objects to an existing group you have two options. If you select the group and the new object and choose the *Group* command, this Designer

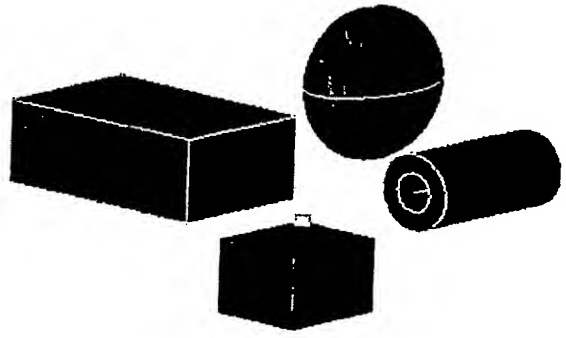
Elements program creates a group within a group. If you want all of the objects to be in a single group, follow these directions:

1. Select the group.
2. Choose ***Layout>Group>Ungroup***. The geometry is ungrouped and the individual objects remain selected.
3. Hold down the SHIFT key and select the geometry you want to add to the group.
4. Choose ***Layout>Group>Group***.

You can follow a similar procedure to remove members from a group.

Grouping Rules

- Grouped objects cannot be extruded, revolved or modified.
- If you change the color of a group and then ungroup the objects, all individual objects retain the new color.
- When grouping objects the *Group* command now ignores objects that only have a point selected. In the graphic below, the three top objects are completely selected and only one control point for the bottom block. When you use the command the control point is ignored. This is helpful when you have many objects located so closely to each other that it might be difficult to select the desired objects without selecting portions of others.



- If you are grouping objects on different layers, the group layer is the current work layer. However, if you ungroup the objects, the objects return to their original layer.

Ungroup Command

The *Ungroup* command separates selected grouped objects or short cuts into their individual components. If the group contains objects that were originally on different layers than the group layer. When ungrouped, the objects return to their original layer.

Using the Ungroup Command

1. Select the group.
2. Choose ***Layout>Group>Ungroup***.

Lock Command

You can lock any object or short cut. Once it is locked you cannot modify, select, translate, rotate or in any way change the position or shape of a locked object.

Tip:
The lock command not only prevents any changes to the locked object, it prevents them from being selected.

Using the Lock Command

1. Select the objects to be locked.
2. Choose ***Layout>Group>Lock***.

If you want to protect an entire document from modification:

1. Choose ***Edit>Select All***.
2. Choose ***Layout>Group>Lock***.

Selected objects can also be locked by clicking the *locked* box in the Edit Objects dialog box.

Unlock Command

You can unlock any Designer Elements program object or short cut that has previously been locked.

1. Choose ***Layout>Group>Unlock***.
2. Select the locked object by clicking on the object or dragging a selection fence around it. The locked object is unlocked and selected.

Arrange Command

The *Arrange* command in the Layout menu allows you reposition overlapping objects. This is a common command for graphics programs and is helpful in editing complicated drawings.

This command displays a submenu of the following commands: *Move Forward*, *Move to Front*, *Move to Back* and *Move Backward*.

Move Forward — Moves the selected object up one position in the display.

Move to Front — Moves the selected object to the top (or in front) of all other objects in the display.

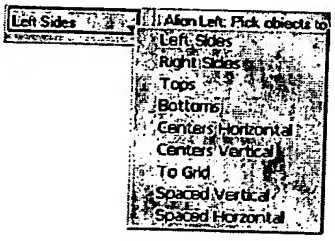
Move to Back — Moves the select object to the back of the display.

Move Backward — Moves the selected object down one position in the display.

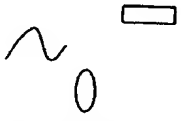
Align Command

This command in the **Transformation** tool palette moves selected objects, including text, relative to other objects.

The *Align* command includes a submenu with these options, *Left Sides*, *Right Sides*, *Tops*, *Bottoms*, *Centers Horizontal*, *Centers Vertical*, *To Grid*, *Spaced vertical* and *Spaced Horizontal*.



The following figure illustrates each of these options.



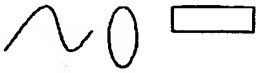
Left Sides — Aligns the left sides of selected objects.

Right Sides

Aligns the right sides of selected objects.

Tops

Aligns the tops of selected objects.

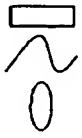


Bottoms

Aligns the bottoms of selected objects.

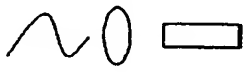
Centers Horizontal

Aligns the center points of selected objects to the same x location.



Centers Vertical

Aligns the center points of selected objects to the same y location.



To Grid

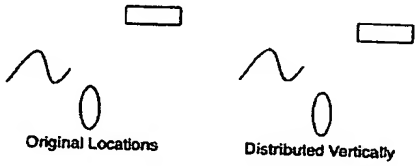
Aligns the left and bottom portion of an object to the closest grid point. The grid does not have to be displayed to use this option.

Spaced Vertical

Equally distributes the selected objects relative to each other.

Spaced Horizontal

Equally distributes the selected objects relative to each other.



Using the Align Command

1. Select the objects to align.
2. Then choose the alignment tool from the Transformation tool palette. The Message Line reads: *Enter Alignment Point.*
3. Choose the point at which you want the objects to align.

Object Commands

These commands allow you to modify various aspects of your geometry including direction, type and size.

Change Direction

This command in the Edit menu reverses the orientation of curves and surfaces.

You may sometimes find it necessary to change the direction of curves to straighten twisted surfaces. Changing the direction of surfaces reverses their normals which can change the way rendered surfaces respond to lighting.

The graphic below shows the direction of a surfaces using the ***Edit>Change Direction*** command. The right graphic shows the changes made by using *Change Direction*.

Using the Change Direction Command

1. Select the object whose direction you want to change.
2. Click ***Edit>Change Direction***.

This Designer Elements program displays temporary arrows along the object to show its direction. You can pick ***View>Redraw Screen*** to make disappear.

Referral:
This command can impact the rendering of your object. See Rendering Options and Flip normals in Chapter 33 for more information.

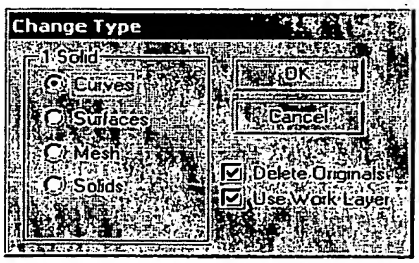
Change Object Type

This command in the Edit menu enables you to change an object's type. This is helpful for editing as well as exporting geometry to software that may not support a particular type of entity. After using this command for a particular operation, this Designer Elements program remembers the selected conversion type (Curves, Surface, Mesh or Solids for a Solid object) until you change the option type.

With this command you can convert:

- Solids to meshes, surfaces or curves
- Surfaces to meshes, surfaces or curves
- Curves to lines, polylines or splines

When you select this command a dialog box similar to the one below appears:



The dialog box contains the following standard options:

Selected Object section — This section contains the conversion options. The options differ depending on whether you select a wireframe, surface or solid object.

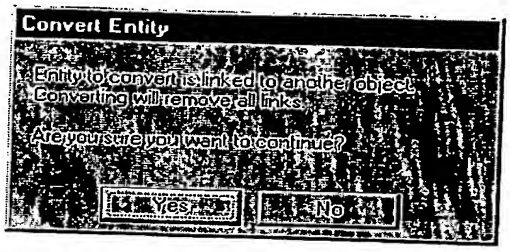
Delete Originals — Selecting this box deletes the original objects following the object conversion.

Use Work Layer — Selecting this box places converted objects on the work layer.

OK — Click this button to convert the object.

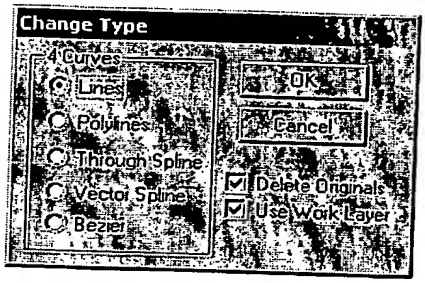
Cancel — Click this button to Cancel the command.

If you are converting a model containing links before the conversion proceeds, the following dialog box displays asking you to confirm your awareness that the links will be removed by completing the conversion.



Changing Curves

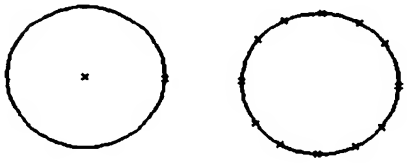
When you choose the *Change Object Type* command after selecting a curve this dialog box appears:



You have the option to convert your curve to Lines Splines. If you want to delete the original curve click the *Delete Originals* check box.

Curves to Lines Option

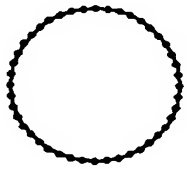
When the *Lines* option is selected you can convert a curve or set of curves to lines. You can use this command to change a spline into a collection of lines or smart polygons into individual lines.



If the *Delete Originals* option is checked this Designer Elements program removes the parent curve after exploding it, leaving only the curves.

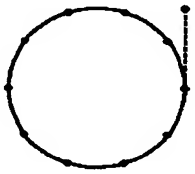
Curves to Polylines Option

When the *Polylines* option is selected, you convert curves or a set of curves into polylines. The graphic here is an example of a circle converted into a polyline. Unlike the line option which would convert a circle into multiple line segments, this option converts the circle into one polyline.



Curves to Through Spline Option

When the *Through Spline* option is selected you convert a line, conic, arc, circle, or ellipse into a Through spline. A Through spline's shape, slope and control points can be edited. This Designer Elements program fits the resulting spline to within a 0.001 drawing unit tolerance of the original curve. If the *Delete Originals* option is checked, this Designer Elements program removes the parent curve after exploding it, leaving only the splines.The graphic here is an example of a circle converted into a Through spline.



Curves to Vector Spline Option

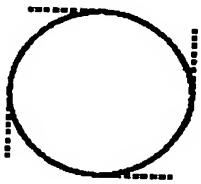
When the *Vector Spline* option is selected you convert a line, conic, arc, circle, or ellipse into a vector spline. A vector spline's shape, slope and control points can be edited. This Designer Elements program fits the resulting spline to within a 0.001 drawing unit tolerance of the original curve. If the *Delete Originals* option is checked, this Designer Elements program removes the parent curve after exploding it, leaving only the splines.

The convert curve command is also useful for removing excessive numbers of control points from constrained splines. The left graphic below shows a spline containing 50 points. The *Change Object Type* command reduces that number to five while maintaining a tolerance of 0.001 inches shown in the right graphic.

After selecting a type click OK and the object will convert.

Curves to Bezier Option

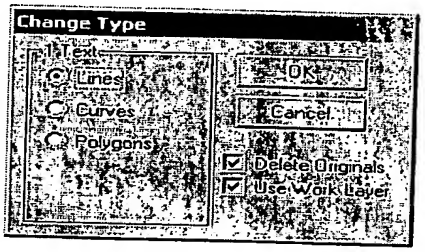
When the *Bezier* spline option is selected you convert a line, conic, arc, circle or ellipse into a bezier spline. The bezier splines shape can be edited by moving the control points and the slope of the spline at those points. If the *Delete Originals* option is checked, this Designer Elements program removes the parent curve after exploding it leaving only the splines.



Converting Text to Curves

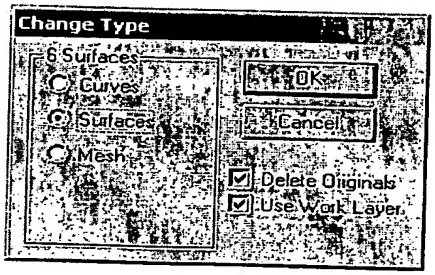
1. Create text using one of the Text tools.
2. Choose ***Edit>Change Object Type***.
3. In the Change Type dialog box choose curves.
4. Click OK.

Each letter of the text converts into a closed grouped curve. You can now perform any curve operation on the text curve.



Changing Surfaces

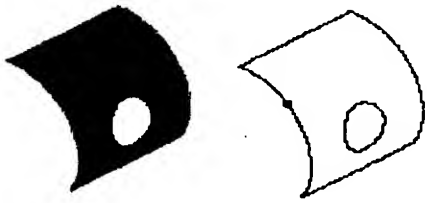
When you choose the *Change Object Type* command after selecting a surface, this dialog box appears:



Surface To Curves Option

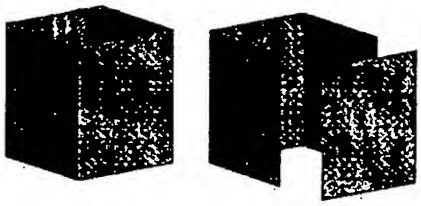
If you select the *Curves* option this Designer Elements program converts the surface into a collection of editable vector splines. You can modify the shape, slope and location of the spline control points. If the *Delete Originals* option is checked, this Designer Elements program removes the parent surface after exploding it leaving

only the curves. The resulting curves may consist of lines, arcs, circles, ellipses or splines.



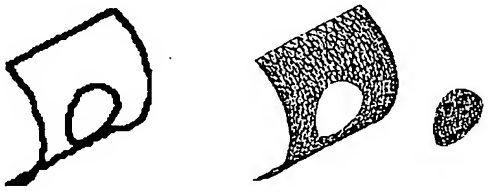
Surface To Surfaces Option

Selecting the *Surfaces* option explodes a surface composed of multiple faces into discrete surfaces. Once the surface is exploded you can edit individual surfaces; move them to a different layer, transform or remove them.



Surface To Mesh Option

If you check the *Mesh* option the surface is converted to a triangular mesh. Meshes are useful for sharing this Designer Elements program data with applications that cannot import precise surfaces. Exploding surfaces to mesh allows you to edit individual vertices.



The precise mathematical representation of a solid or surface must often be converted into a collection of imprecise planar facets. These facets, for example, may be used to export a model to the STL format, and when changing a solid or surface

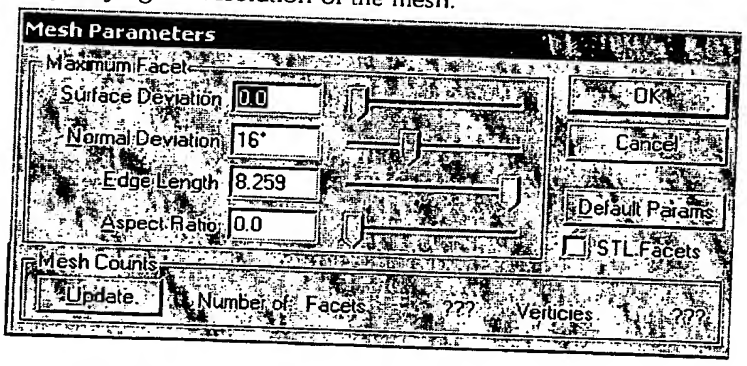
(Edit>Change Object Type) to a mesh. The amount error that results from this conversion is controlled by the settings in the mesh parameters dialog box.

During the conversion vertex points are distributed on the surface or solid. These vertices are then grouped into 3-sided and 4-sided facets. The conversion is deemed acceptable when the generated vertices and facets satisfy the settings. The five available settings are: Surface Deviation, Normal Deviation, Edge Length, Aspect Ratio and STL Facets. These settings are defined in the sections below.

Change the facet settings as needed in the dialog and then click the Update button to see the number of facets and vertices generated. Determining the combination of settings that will work for a given situation can be a little bit of an art. If one setting becomes too tight the other settings will have no effect. If one setting becomes too loose, it will have no effect.

Keep in mind that under "real" circumstances the settings are used by the faceting algorithms if possible. It is often not possible to satisfy all settings simultaneously. In this situation the algorithm decides which settings to "loosen".

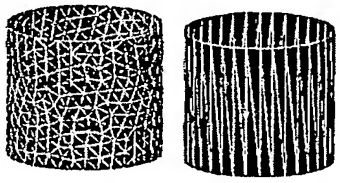
When you explode a surface into a mesh, the mesh dialog box appears with characteristics for specifying the resolution of the mesh.



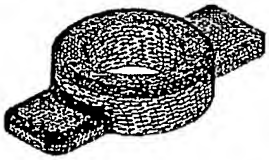
The dialog box includes:

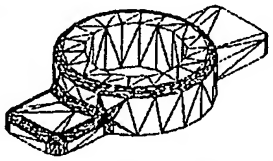
Aspect Ratio — Sets the maximum ratio between triangle edges. It prevents forming long narrow triangles. Enter a value in the field or use the slide to set the value.

Surface Deviation — Sets the maximum acceptable distance between the facet and the surface represented by the mesh.

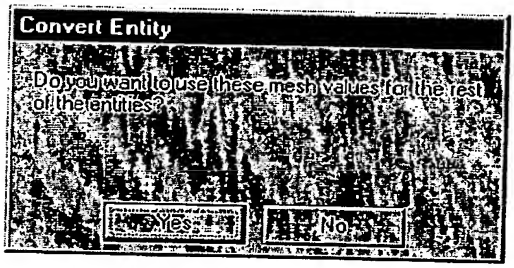
	Enter a value in the data field or use the slide to set the value.
Normal Deviation	Sets the maximum angular deviation between adjacent facets. 20 degrees is the default. Use 10 for extremely dense meshes. Enter a value in the data field or use the slide to set the value.
Edge Length	Sets the maximum acceptable edge length for facets. Enter a value in the data field or use the slide to set the value.
Mesh Count	This section includes the Update button and information on the Number of Facets and Vertices. Click the Update button to calculate the approximate number of facets based on mesh characteristics. The Number of Facets and Vertices information display questions marks until you click the Update button. The calculated values replace the question marks.
Smoothing	Selecting this option creates facets that are regularly spaced. The left graphic here shows an object without smoothing. The right shows the object with mesh smoothing.
	
Default Params	Click this button to revert the mesh parameters to this Designer Elements program default values.
STL Facets	This setting will force the facets generated to be suitable for stereolithography usage. This setting is usually used when exporting STL files.

The mesh in the left graphic below was created with a *Normal Deviation* of 10 and a *Max Edge Length* of 0.1. The right graphic was created with a *Normal Deviation* of 20 and a *Max Edge Length* of 0.





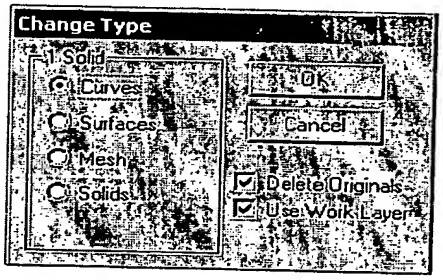
After setting the desired values, click OK and the object will convert. If you select more than one surface to convert the following dialog box appears:



Click Yes to use the same mesh conversion values for all selected surfaces.

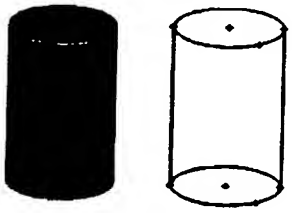
Changing Solids

When you choose the *Change Object Type* command after selecting a solid this dialog box appears:



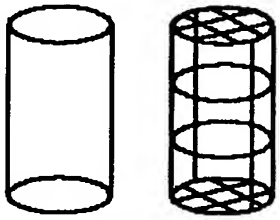
Solid To Curves Option

If you select the Curves option, this Designer Elements program converts the solid into a collection of editable vector splines. You can modify the shape, slope and location of the spline control points. If the *Delete Originals* option is checked, this Designer Elements program removes the parent solid after exploding it leaving only the curves. The resulting curves may be lines, arcs, circles, ellipses or splines.



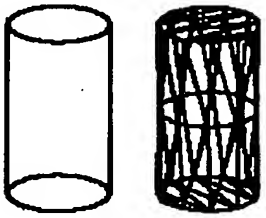
Solid to Surfaces Option

Selecting the Surfaces option explodes a solid into discrete surfaces. Once the solid is exploded you can edit individual surfaces, move them to a different layer, transform or remove them.

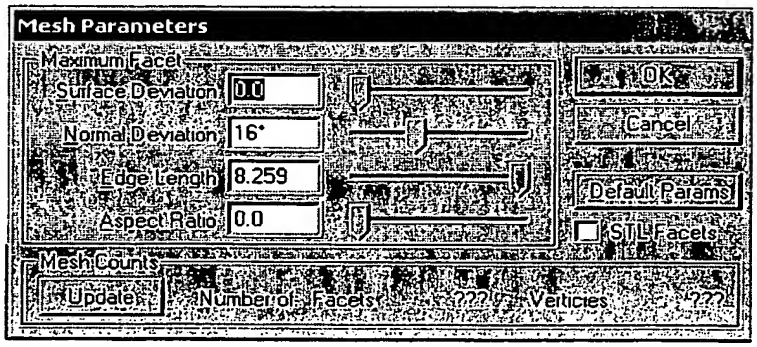


Solid To Mesh Option

If you check the *Mesh* option the solid converts to a triangular mesh. Exploding solids to meshes allows you to edit individual vertices.



When you explode a solid into a mesh, the mesh dialog box appears with characteristics for specifying the resolution of the mesh:

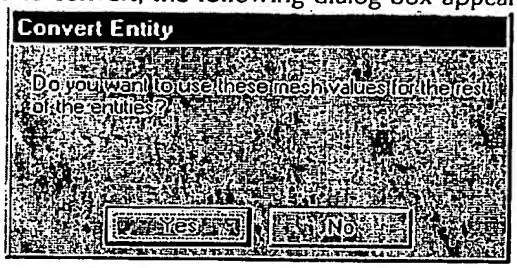


The dialog box includes:

Aspect Ratio — Sets the maximum ratio between triangle edges. It prevents forming long narrow triangles. Enter a value in the data field or use the slide to set the value.

Surface Deviation	Sets the maximum acceptable distance between the facet and the surface represented by the mesh. Enter a value in the field or use the slide to set the value.
Normal Deviation	Sets the maximum angular deviation between adjacent facets. 20 degrees is the default. Use 10 for extremely dense meshes. Enter a value in the data field or use the slide to set the value.
Edge Length	Sets the maximum acceptable edge length for facets. Enter a value in the data field or use the slide to set the value.
Mesh Count	This section includes the Update button and information on the Number of Facets and Vertices. Click the Update button to calculate the approximate number of facets based on mesh characteristics. The Number of Facets and Vertices information display questions marks until you click the Update button. The calculated values replace the question marks.
Smoothing	Selecting this option creates facets that are regularly spaced. (See the graphic included with the Surface to Mesh conversion in an earlier section for a smoothing effect example.)
Default Params	Click this button to revert the mesh parameters to this Designer Elements program default values.
STL Facets	This setting will force the facets generated to be suitable for stereolithography usage. This setting is usually used when exporting STL files.

After entering the desired values, click OK and the object will convert. If you select more than one solid to convert, the following dialog box appears:



Click Yes to use the same mesh conversion values for all selected solids.

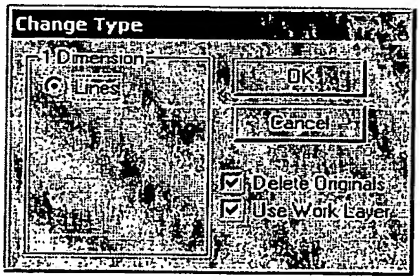
Solids To Solids Option

You can use the Solids option to break one solid up into several smaller units. This option comes in handy if you want to edit only one part of a larger solid.

If you want to find out the mass properties of several solids as if they are combined into one, you can first unite them and analyze their properties. After your measurement is completed use the *Change Object Type Solid* option to break the part back into individual pieces.

Change Dimensions

When you choose the *Change Object Type* command after selecting a dimension this dialog box appears.



Use this to change dimensions into lines and text. This is helpful when exporting files with dimensions that may not be supported by the other software.

Simplify Object

This command in the Edit menu will examine the selected objects and if the entity falls within tolerances, will do the following:

- Convert zero length curves to points
- Convert circular splines to arcs or circles
- Convert straight splines to lines

- Convert NURB surface or solid faces to analytics

Potential NURB shapes converted to analytic shapes include cylinders, cones, speres, tori and planes. Candidate shapes for simplification include data imported via IGES, Step, Rhino and Alias. Also some surfaces created in this Designer Elements program are initially created as NURBS and may simplify to an analytic shape.

Some advantages of analytic representation of shapes over NURBS are analytic operations are considered exact whereas NURBS are precise. Analytic operations execure much faster internally and take up less memory in RAM and less space when the file is saved. For example, an IGES file can be reduced by half after using the Simplify Object command.

Note: Using the Simplify Objects command may cause loss of part history on complex objects.

Edit Objects

This command in the Window menu edits selected objects by changing individual characteristics such as: weight, layer, resolution and other specifications.

You can display the Edit Objects dialog box in two ways:

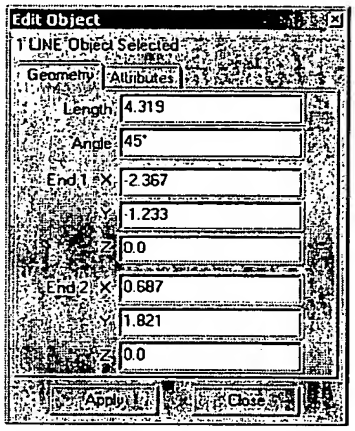
- Select the object and choosing ***Window>Edit Objects***
- Double-click on the object.

Note: You cannot double-click on editing operations performed on a solid such as blending, placing holes, shelling etc. except from within the Design Explorer **(Cobalt™ and Xenon™ only)**. (See a later section for information on the Design Explorer.)

Changes made through this dialog box can be reversed with the *Undo* command.

When you choose Edit Objects a dialog box similar to the one here displays containing between two and four possible tabs: *Geometry, Display, Attributes* and *Material.*

Geometry Tab



All tools that create geometry will include a geometry tab in the Edit Objects dialog box.

The specific characteristics listed depend on the object selected. The object selected in this case was a line and includes the following characteristics: *Length, Angle, End 1* (X, Y and Z values) and *End 2* (X, Y and Z values). All values appear in this dialog box as whole numbers and decimals, depending on the decimal setting in Preferences.

Entering Values

You can enter values in this dialog box in any units desired. These will automatically be converted into the units you set in Preferences. See Chapter 6.

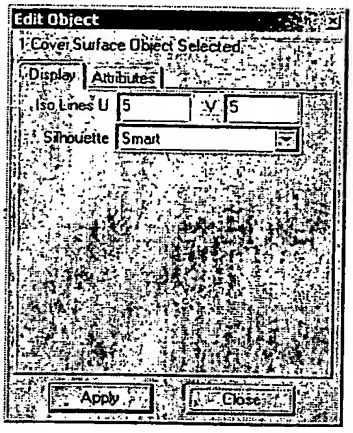
When units are set to feet and inches (in the Prefernces dialog box) it's important to be aware of the following rules:

- All numbers are assumed to be feet unless accompanied by the unit symbol like " for inches. Entering a 1.5 in the field is read as 1.5 feet. If you want 1.5 inches, enter 1.5", 1.5i, 1.5in, 1.5 inch, etc.
- If you want to enter fractions of inches, each entry must include the unit symbol. For example, 5 feet 6 5/8 inches must be entered 5 6" 5"/8. Internally this is converted as 5' + 6" + 5"/8. If the inch symbol is not included with the fraction, 5/8 will be interpreted as a fraction of a foot.

Drawing views created for Layouts have their own Geometry page. Some editing operations like *Split Solid* contain no specific geometry characteristics and therefore do not have a Geometry page.

Every chapter in this manual that deals with this Designer Elements program objects includes a *Geometric Characteristics* section listing all characteristics specific to the object. Those features that do not have specific characteristics are noted accordingly.

Display Tab



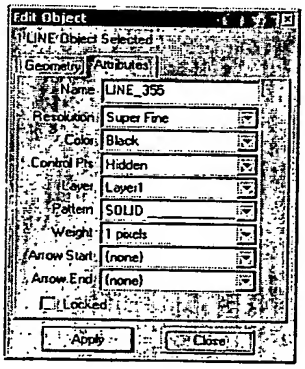
This page in Edit Objects appears for surface and solid objects only.

The graphic shows the characteristics included in the Display page for a surface object and includes: *Iso Lines* and *Silhouette*.

The number of Iso Lines allows you to control the isopram lines drawn for a surface. These Iso (isopram) lines are constant parameter curves that lie on a surface, typically defined in parameter space. The parameter space coordinate system uses *U* and *V* coordinates, where U = horizontal and V = vertical. A zero (0) in both fields turns off Iso lines. The appropriate *U/V* values may enhance the visual appearance of the surface at the expense of drawing speed. The letters *U* and *V* are industry standard space coordinate identifiers.

The *Silhouette* option controls whether this Designer Elements program displays silhouette edges of objects. Silhouette edges are view dependent and can cause a significant reduction in drawing speed. The pull-down menu displays the options: On, Off and Smart. Smart Silhouettes display a silhouette only if it does not degrade the display performance.

Attributes Tab



This tab displays a standard list of characteristics for the selected object.

The attributes include:

Name — Displays the name of the object. You can enter a new name if you wish.

Resolution — Displays the resolution setting for the object. You can select any of the

options listed in the pull-down menu. See the Object Resolution section for more information.

Color — Displays the color of the selected object. You can choose any color for your object.

If you object is on a layer with an override color, the object will display in the override color but the Edit Objects color field will display the actual color of the object.

Control Pts — Displays the status of the control points, hidden or visible. You can choose either setting.

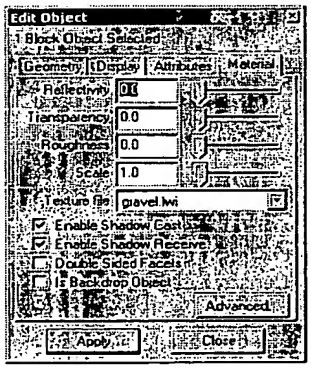
Layer — Displays the layer where the object is located. You can place the object on any available layer.

Locked — Choosing this option locks the object preventing it from being selected or modified. This is same operation that occurs when choosing ***Layout>Group>Lock***.

For wireframe objects this tab also includes: Pattern, Weight, Arrow Start and Arrow End.

Material Tab

This tab appears for objects that have rendering materials applied to them.



The material characteristics include the following:

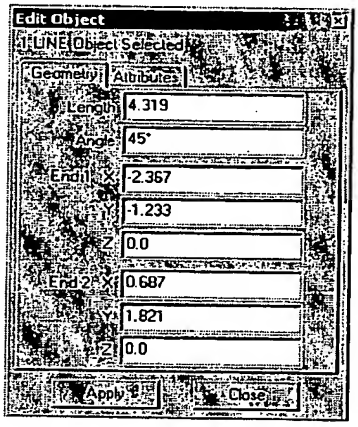
Reflectivity — This field sets the material's reflectivity. Values can be entered between zero (0) and one (1). Entering a zero in the field renders a flat finish. Entering a one in the field renders a mirrored finish.

Transparency	This field sets the transparency of the material. Values can be entered between zero (0=transparent) and one (1=opaque). This option only works correctly when objects are rendered with ray trace rendering commands.
Roughness	This field sets the roughness of the material. Values can be entered between zero (0) and one (1).
Scale	This field sets the scale of the material. Typically scale increases the size of the detail. Values can be entered equal to or greater than zero (0).
Texture File	This field displays the selected material. The pull-down menu lists all images in the texture folder. For those materials that do not support textures this menu is not available.
Enable Shadow Cast	This check box allows you to specify if an object casts shadows. This is valuable for reducing the shadows present in complex drawings. When checked the selected object casts shadows.
Enable Shadow Receive	This check box allows you to specify whether an object will receive shadows from other objects. This ability is valuable for reducing the number of shadows present in complex drawings. When checked the selected object receives shadows.
Double Sided Facets	This check box provides you with additional rendering control for surfaces. Objects with normals facing away from the line of sight are not rendered. When checked all objects are rendered regardless of the normal direction. Select this option for objects that cause light refraction like glass. When left unchecked facets on the back side of the glass are ignored resulting in an inaccurate rendering.
Is Backdrop Object	This check box allows you to optimize rendering calculation time for an object like a wall, that functions only as a backdrop for other objects. Since

the object automatically receives light due to its large size, this Designer Elements program does not need to spend much time performing light ray calculations.

Advanced button — Clicking this button displays the Render Material Settings dialog box. Use this dialog to change the advanced rendering settings used on the selected object. See Chapter 33 for a detailed description of these settings

Using the Editing Objects Command



1. Select an object to edit.
2. Choose ***Window>Edit Objects*** and the dialog box appears.

   The dialog box contains between two or four possible tabs depending on the object you select, *Geometry*, *Display*, *Attributes* or *Materials*.
3. Click on the tab containing the desired information.
4. Change the information in an entry box by double-clicking the entry data field and typing a new entry. Do not press ENTER (Windows) or RETURN (Macintosh). Many data fields have pop-up menus for selection. Press the down arrow and then drag to the selection you want.
5. Make changes in other entry fields as required.
6. Click Apply and the changes automatically occur on your selected geometry.
7. Click Close to close the Edit Objects dialog box.

You can use *Undo* or *Redo* to reverse changes made through this dialog box.

Coordinates are measured in the units set in the *Units* page of Preferences.

If you are doing a lot of editing you may want to leave the Edit Objects dialog box displayed. That way you can select the object, make the changes in the dialog box, click Apply and then go on to the next object.

If more than one object is selected when you choose *Edit Objects,* only the common information is displayed. Entries are blank when the information isn't common. For example, if you select two concentric circles, the center will be displayed in the Edit Objects box but the diameter box will be blank.

If you select several objects and then choose *Edit Objects*, all of the objects will reflect the changes you make in the dialog box. For example, if you select all dimensions and change the text entry in the Edit Objects dialog box to 2, all the dimensions will display a 2.

Moving an Object to a Different Layer

1. Select the object(s) to be moved.
2. Choose ***Window>Edit Objects.***
3. Click the arrow for the Layer entry field.
4. Pick the layer from the list.
5. Choose Apply.

Links

Many of the objects in this Designer Elements program are associated with other objects in parent-child relationships. Associative objects retain a history of how they were created and a set of rules that define their geometric description. If you modify a parent object (an object that defines another object), this Designer Elements program automatically regenerates the dependent or child object. For example, consider a skinned surface that is defined by two lines. If you modify one of the lines, the skinned surface will automatically regenerate.

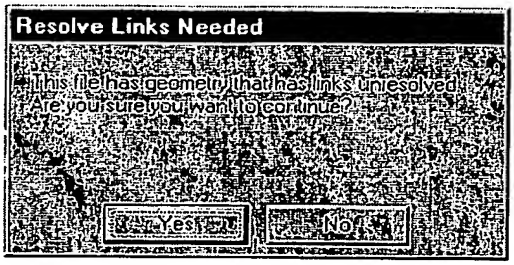
This Designer Elements program provides two commands for dealing with Links: *Resolve* and *Remove.*

Resolve Links Command

This command is in the Edit menu. As you modify an entity that is part of a parent-child relationship the word, *Edit* in the menu bar and the command, *Resolve Links* highlight in red.

In most cases, this Designer Elements program will resolve the links in a parent/child relationship automatically and the words will un-highlight when finished. If the words stay red, select ***Edit>Resolve Links*** to force the program to resolve the relationship.

If you attempt to save a file containing links that are not resolved, the following warning appears:



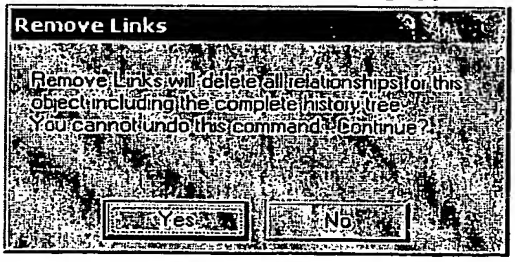
Remove Links Command

The *Remove Links* command in the Edit menu deletes all associative links between a parent and child.

Referral/Tech Note:

Do not use the *Remove Links* command to remove a feature from the history tree. This removes all parent/child associativity for your geometry. Use the Design Explorer instead. See a later section for information on the Design Explorer.

When you select the command the following warning appears:

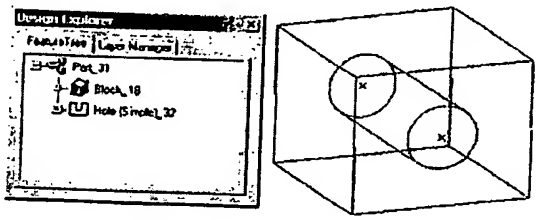


This warning reminds you that the selected object has a parent/child relationship to another object and that deleting this object not only deletes the history tree but cannot be undone.

Press the Yes button to continue the *Remove Links* command or No to discontinue.

Design Explorer

This Designer Elements programs Design Explorer is a valuable feature for traversing your drawing's design history. It offers a Windows-style tree structure with expandable branches that display, in a hierarchical fashion, the associative geometry used in the creation of all of the entities in the drawing.



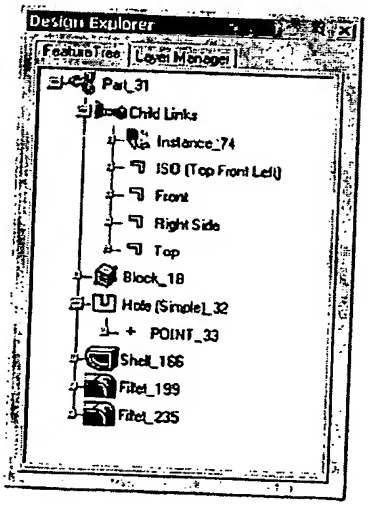
The figure shows a solid cube with a hole in one face. The Design Explorer window identifies the entity as a part and shows that it consists of a block, the cube and a hole feature. As you click on items in the tree the corresponding part in the drawing will highlight.

If you select an item in the Design Explorer when the Edit Objects dialog box is displayed, the dialog box automatically updates to reflect the selected item.

Child Links

The child links feature organizes all persistent parts of your model in the Design Explorer. If you place material or decals on your model or create drawing views they group separately from any geometry operations under the *Child Links* category.

This frees you up to modify your part without having to replace material or recreate views.



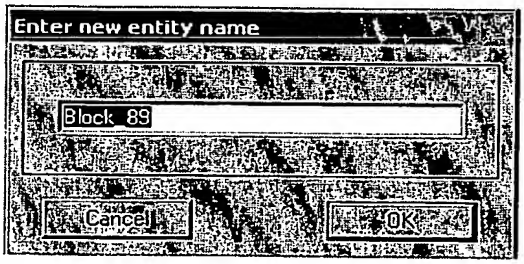
Cobalt and Xenon Only

Design Explorer Commands

The Design Explorer also includes several of its own menu of commands. For Windows users, (and Macintosh users with a two button mouse) click the right mouse button on the feature name to display the menu. For Macintosh users, hold down the CONTROL key and click on the feature. Depending on where you right click and when, one of the following menus appear:

Rename

The *Rename* command lets you rename any of the entities. It displays this dialog box.



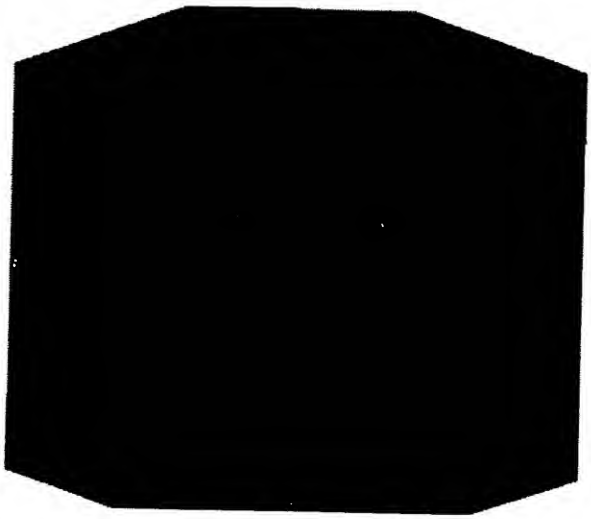
Enter the new name and click OK.

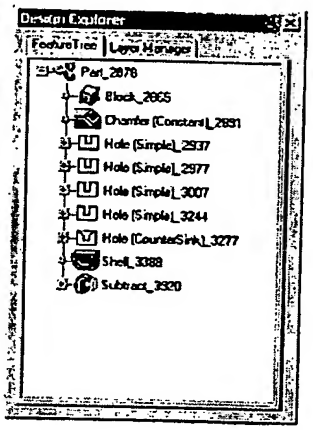
Suppress Feature/Unsuppress Feature

These commands allow you to turn off or on a particular feature in the solid history tree. They toggle between each other depending on whether an operation is suppressed.

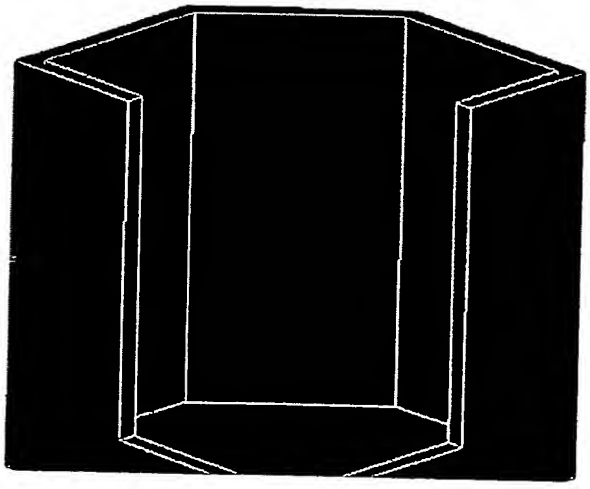
When a feature is suppressed, this Designer Elements program rebuilds the solid as if the operation never occurred. This feature can be unsuppressed and reintegrated

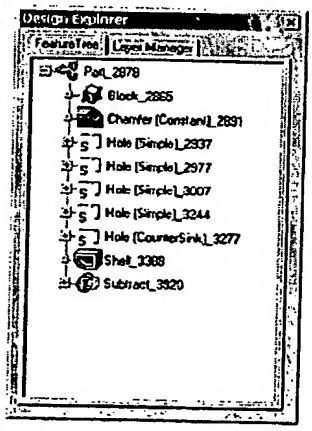
into the part at a later time. The graphic below shows a part and the Design Explorer open.





This next graphic shows the same part with all of the holes suppressed. Notice the "S" displayed over the operation icon in the Design Explorer.





Some uses for feature suppression include the following:

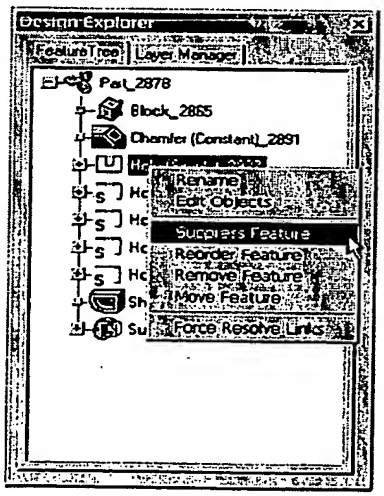
- FEA Models: when creating FEA (Finite Element Analysis) models you may not want certain features included in the analysis.
- Part Regeneration: removing certain features may speed up rebuilding the part, including display time.
- Shelling: if a part fails to shell you can turn off blends and other features that may prevent a successful shelling operation. Then, using the *Reorder Feature* command, you can move the shelling operation before the suppressed feature in the history tree so that shelling succeeds. Finally, you can unsuppress the selected feature using the *Unsuppress Feature* command.

Using the Suppress Feature Command

1. Select the model which contains the feature you want to suppress.
2. Choose ***Window>Design Explorer*** to display the history tree.
3. Expand the tree to display the desired feature by clicking on the plus (+) sign to the left of the part name.
4. Select the desired feature.

   The feature highlights.
5. Click the right mouse button (Windows) or press the CONTROL key (Macintosh) to display the Design Explorer menu.
6. Choose the *Suppress Feature* command.

   The feature suppresses and your model regenerates without the feature. The Design Explorer now displays an "S" over the operation icon.

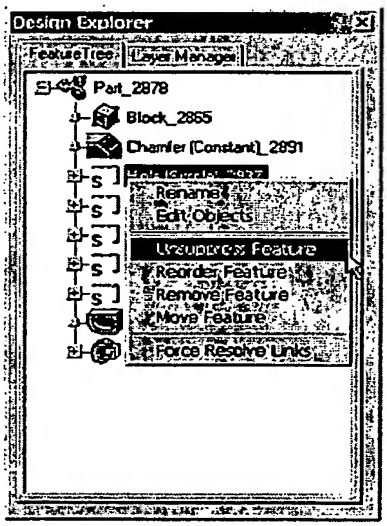


Using the Unsuppress Feature Command

The *Unsuppress Feature* command replaces the *Suppress Feature* command in the Design Explorer menu for suppressed features only.

1. In the Design Explorer select the suppressed feature.
2. Click the right mouse button (Windows) or press the CONTROL key (Macintosh) to display the Design Explorer menu.
3. Choose the *Unsuppress Feature* command.

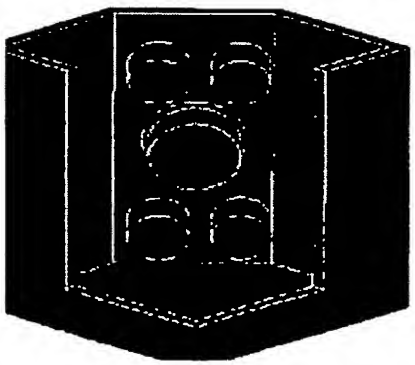
   The feature reappears and the model regenerates. The feature icon no longer displays an "S."



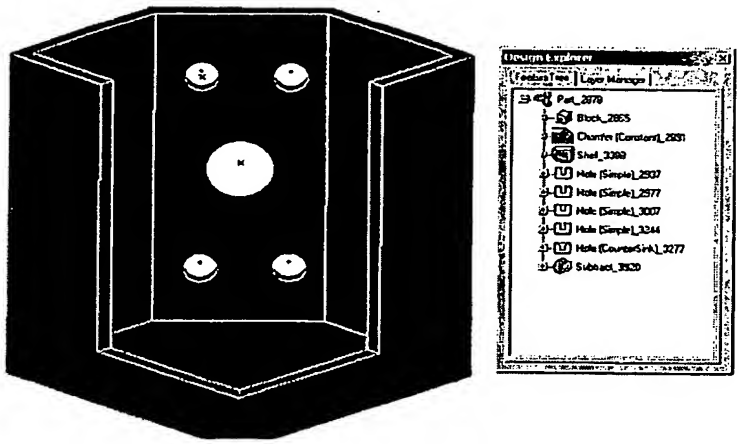
Cobalt and Xenon Only

Reorder Feature

Sometimes you may need to modify your part by rearranging its features. This command allows you to move features up or down in the history tree. In the graphic here, the shell operation occurred after the holes were placed.



Using the *Reorder* command, the shelling operation was moved before the hole operation, resulting in the model shown here.



Cobalt and Xenon Only

Using the Reorder Feature Command

1. Select the part.
2. Open the Design Explorer and display the tree to expose the feature you want to reorder.
3. In the Design Explorer menu select the *Reorder Feature* command.

   The Message Line reads: *Feature Reorder: Click on new location in Design Explorer.*
4. Click on the operation below which you want to move the selected operation.

   The operation moves and the part regenerates.

Move Feature

The *Move Feature* command allows you to reposition holes and bosses within the assembly.

Using Move Feature Command

1. Select the feature in the Design Explorer that you want to move. The feature is selected in your drawing area.
2. Display the Design Explorer menu (Right mouse button - Windows, CONTROL+mouse button - Macintosh).

Tech Note/Tip:

You cannot move or reorient a feature to any side of a solid that is not visible. The operation automatically believes you are choosing one of the visible faces. Rotate the solid so that the face is visible before choosing the Move Feature command.

3. Select the *Move* command. The Message Line tells you to select a new center for the feature.
4. Click a new center on your solid. The feature is repositioned.

Remove Feature

This command deletes a feature from the assembly.

Edit Parameters

This command displays the Edit Objects dialog box. You can also double-click any item in the tree to open Edit Objects.

Expand Item

This commands expands the part of the tree under the selected item.

Collapse Item

This command collapses the part of the tree under the selected item.

Cobalt and Xenon Only

Expand Branch

This command expands the entire branch that includes the selected item.

Collapse Branch

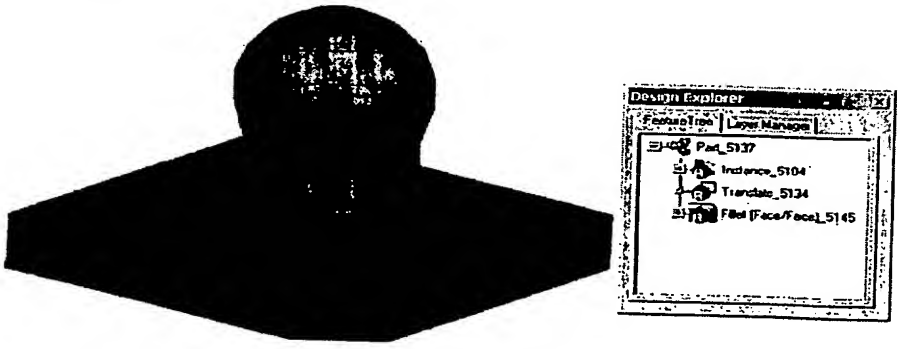
This command collapses the entire branch that includes the selected item.

Close Window

This command closes the Design Explorer.

Force Resolve Links

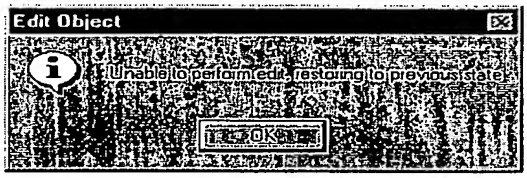
This command resolves all unresolved child geometry. If you select the part and display the Design Explorer, a red "R" appears over any child operation icons that is not resolved.



This will occur if you had turned off the *Enable Auto Resolve* option in the General page of Preferences. When activated, this Designer Elements program automatically resolves all child geometry when a parent is modified. When deactivated use the *Force Resolve Links* command to resolve child geometry.

Design Explorer Rules

- Within the Design Explorer you cannot select a curve within a profile and edit its length through the Edit Objects dialog box. This attempt will make a later operation invalid. The message here appears when you choose Apply in the Edit Objects dialog box.



Cobalt and Xenon Only

  Click OK to restore the original length.

  For example, if you want to lengthen a line that has been extruded to a solid, the change in length does not affect the lines connected to the edited line. The closed profile would be destroyed. To lengthen a line use the Move tool and select the endpoint of the line and move it. By default, it will select and move the endpoints of all lines connected at that endpoint.

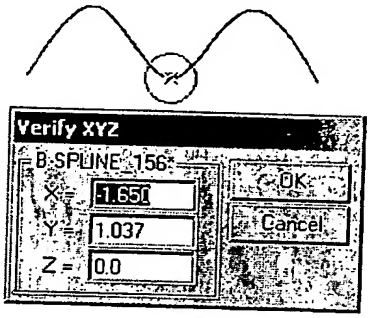
- The Design Explorer will only show one part at a time.

Verify Menu

The Verify menu gives you access to a variety of commands that provides information about your file or about specific objects.

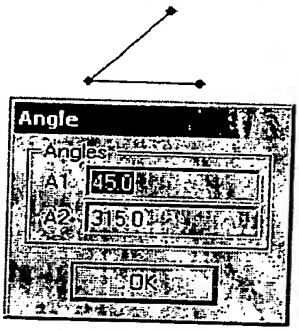
X,Y, Z Command

The *XYZ* command displays the model coordinate values of the selected point. You may edit and change a selected value.



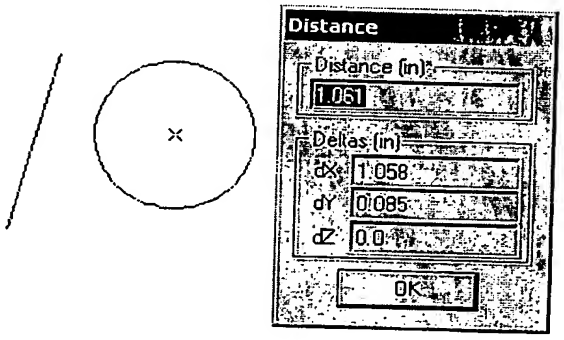
Angle 3 Pts Command

The *Angle 3 Pts* command calculates the angle formed by three points that you choose.



Distance Pt-Pt Command

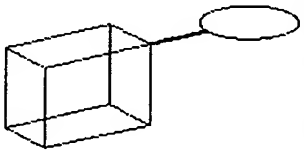
The *Distant Pt-Pt* command calculates the distance between two points taken from the drafting assistant.

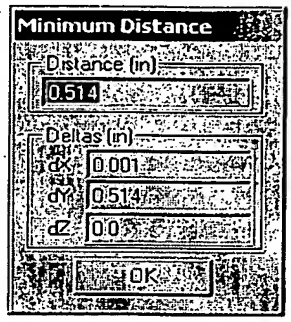


Minimum Distance Command

The *Minimum Distance* command calculates the smallest distance between two selected objects.

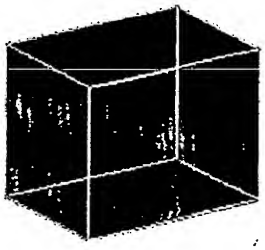
This command is useful for checking if two objects interfere with each other. It will calculate the minimum distance between any combination of curves, surfaces or solids.

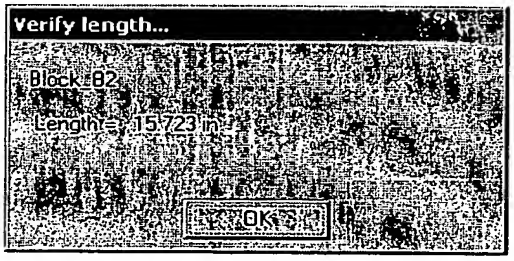




Length Command

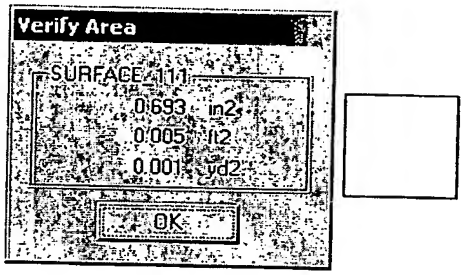
This *Length* command calculates the length of a line, spline, arc, circle, conic, surface and solids. In the case of surfaces and solids the command adds all edge lengths, providing a perimeter length value as shown in the graphic here.





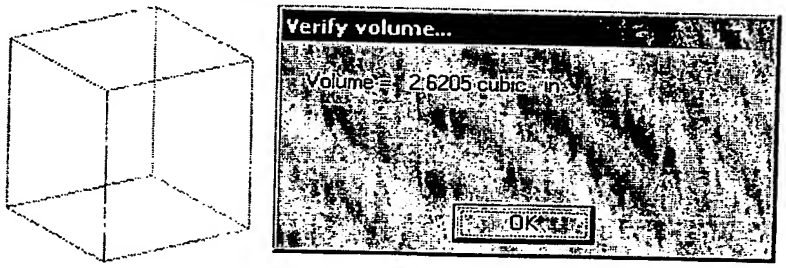
Area Command

The *Area* command calculates the area of closed curves or a surface. These curves include basic objects such as circles and ellipses but also splines connected to create an enclosed area.



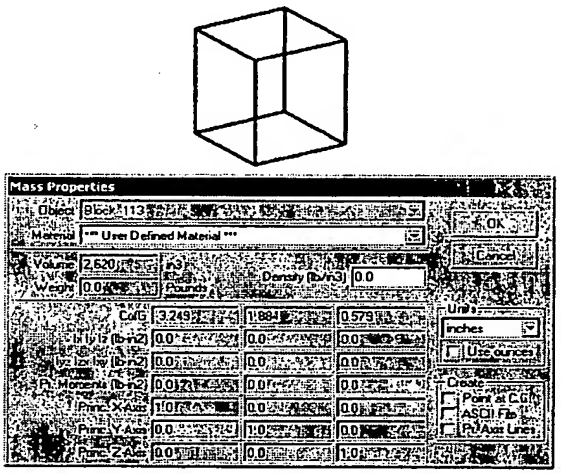
Volume Command

The *Volume* command calculates the volume of a closed object.



Properties Command

The *Properties* command calculates a solid's volume, center of gravity, principle moments, or moments of inertia.



The Properties dialog box includes the following sections:

Material

This pull-down menu allows you to specify a material type.

The menu includes User Defined, (Misc) Cast Iron (typical), Steel (SAE 1020, SAE 4130 normalized, SAE 4140, 18-8 stainless, Aluminum (2024-T3, 2014-T6, 7075-T-6), Magnesium (M-1/Am35, AZ61A/AMC575), (Titanium) Titanium (typical), Wood (Birch grain W, Birch grain X, White Oak grain W, White Oak grain X, Douglas Fir grain W, Douglas Fir grain X, Souther Pine grain W, Southern Pine grain X), Ceramic (Brick Soft, Brick Hard, Sewer Pipe, Glass), Aggregate (Concrete low strength, Concrete medium strength, Concrete high strength).

Tech Note:

This material selected here is set separately from material hatching used for geometry in a section view.

See Chapter 28 for more information on crosshatching and Chapter 36 for more information on section views.

Select one of these materials and the associated density displays in the Density field. For the User-defined material you can enter your own value.

Basic

This section includes the volume, weight and density of the selected object. The density is based on the selected material.

Advanced

This section includes the Center of Gravity, Moments of Inertia and axis information.

Create

This section includes three check boxes: ASCII file, Point at C.G and Pr. Axis Lines.

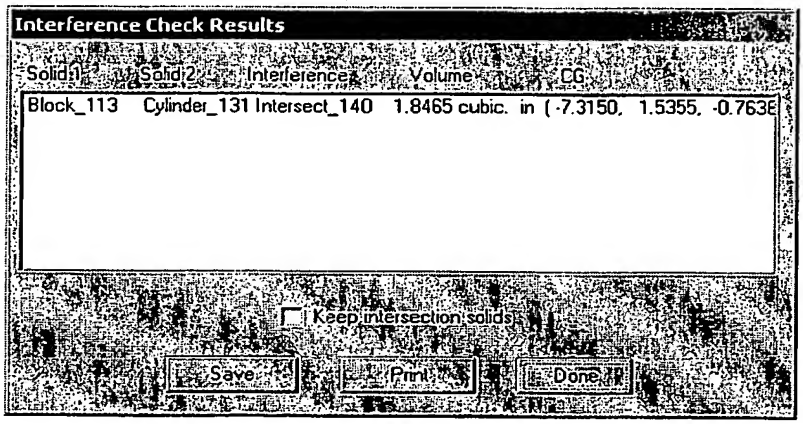
ASCII File - Clicking in the check box allows you to export the information contained in this dialog box to an ASCII file. When you click OK to close the Properties dialog box, the *Save document as* window appears asking for a name and location for the ASCII file.

Point at C.G. - Clicking in this check box, places a point at the Center of Gravity for the selected object. Click OK to close the dialog box and a C. G. point appears. The style of the point is based on the selected style for the **Point** tool. See Chapter 9 for more information.

Pr. Axis Lines - Clicking in this check box, gives you the axis direction vectors for the coordinate system where the products of inertia vanish.

Interference Command

This command in the Verify menu checks the interface volume of two or more solids to determine an intersection or interference. If an interference is found the Interference Check Results dialog box appears:



The dialog box contains the interference list window: *Keep intersection solids* check box, and the Save, Print and Done buttons.

Interference window	This window lists the interference between the selected solids and includes Solid 1 and Solid 2 (the solids names), Interference (the interference solid name), Volume (the volume of the interference solid), and the CG (the center of gravity for the interference solid - x, y and z location).
Keep intersection solids	When checked this option retains the intersection solids in your drawing.
Save	Click this button to save the interference data to a log file which can be opened in any text editor.
Print	Click this button to print the interference data.
Done	Click this button to close the dialog box.

Using the Interference Command

1. Display your model as a wireframe.

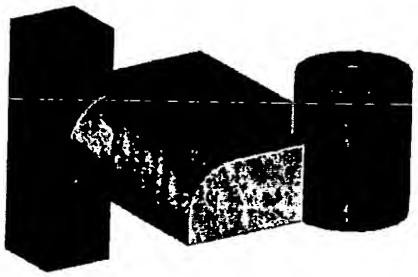
2. Select the solids on which you want to perform the check.
3. Choose ***Verify>Interference***.

   If an interference is encountered the Interference Check Results dialog box appears listing the interference and displaying the interference solids in black on your model.
4. If you want to retain these interference solids in your model check the *Keep intersection solids* box.
5. Click Save or Print if you would like to save or print the log of the interference solids.
6. Click Done to close the dialog box.

   If *Keep intersection solids* box is checked your model displays the intersection solids.

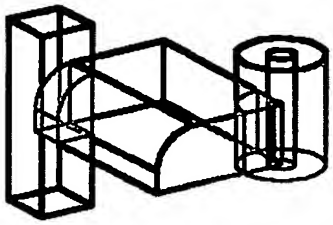
   Example:

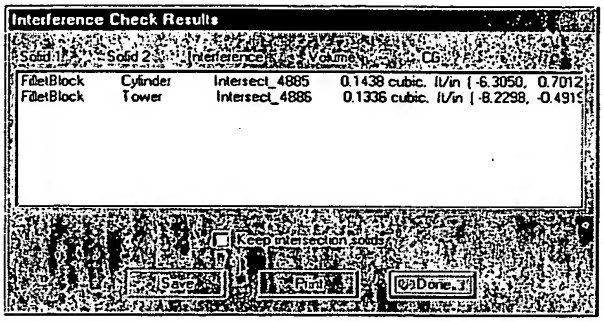
   The model here displays three intersecting solids.



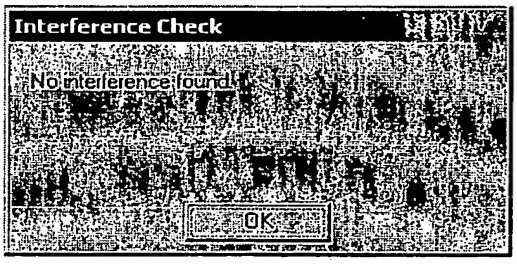
By choosing ***Verify>Interference*** on these selected models (shown below in wireframe), the Interference Check Results dialog box appears and the interfer-

ence solids are shown in black.





If no interference is found the following dialog box appears.

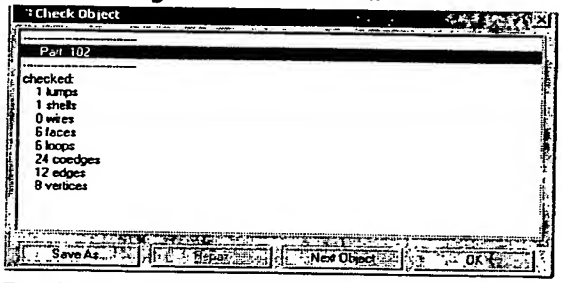


Object Counts Command

This command counts the number of objects in the current drawing.

For grouped objects, each individual object is counted as well as the group itself.

Check Object Command



The Check Object command examines an object for proper data structure, topology and issues geometric warnings associated with ACIS data. This includes checks for curves, surfaces, and solids. The tool is frequently used in conjunction with imported data created from non-ACIS kernels. Some of the many items checked include:

Data Structure Checks

- Entities have has appropriate child-level entity; e.g.: body has lump. Face has edges, etc
- Presence (non-NULL) and closure of back pointer from child to parent; e.g.: body's lump points to body.
- The coedge on spline surface has pcurve.

- Pcurve indexing (0/+-1/+-2) is appropriate.
- The pcurve has non-NULL 2D B-spline curve.
- If edge has non-NULL curve, then curve must have equation.

Topological Checks

- Loops must be closed in both the next and prev directions.
- Apex edge loops are correct.
- Coedge has a partner, except apex coedge
- All coedge partners point to same edge
- Sequential coedges share a vertex
- Edge is in exactly one of start and end vertex edge groups. For example, edge can be reached for 1 value of i using start()->edge(i)->coedge() and partner and next (or previous) pointers.

Geometric Checks

- Face gaps along shared edges and vertices
- Entities with geometry must have non-NULL geometry. For example, a face points to a surface.
- Analyzes a curve or surface for C1/G1 discontinuties
- Self intersecting surfaces
- Analytic surfaces have valid definitions
- Pcurve surface matches face surface (warning only if not equal since surface could be trimmed).
- Pcurve form is agrees with curve form, e.g. closed, open, periodic.
- Pcurve parameter period agrees with curve period.
- Pcurve at points 0, 1/3, 2/3, and 1 way along curve must lie on the edge and tangent directions at these points must roughly agree, i.e., have positive dot product. This also tests the following:
- Spline surface form is set correctly, e.g. surfaces closed in u report this. Checks the underlying 3D B-spline surface at 10 points along seam to verify form.
- Checks that coedge vertices do not lie on spline surface singularities.

- Face normal is consistent with coedge direction.
- Start and end vertices of coedge lie on face.
- Edge lies on face. Checks at 10 points along edge.
- Start and end vertices lie on edge geometry.
- Faces are ordered correctly around edge, according to sidedness.
- Coedges are ordered correctly around edge, according to face curvature.
- Edge has same sense as curve (taking reverse bit into account).
- Checks curve has correct form.
- Edge parameter range is good and agrees with start and end points.
- Check edge for bad approximation direction.
- D3 checks on intcurve (option that can be turned on).
- No two vertices have the same location.
- Optional face/face intersection checking (option check_ff_int).

You may select one or more objects with the Check Object command. A report is presented in the above dialog box. The dialog box has the following options:

- Save As: Saves the reported as an ASCII file.
- Repair: Attempts to repair any errors that were reported.
- Next Object: Step to the next item in the list.
- OK: Exit back to the program.

Transforming Geometry

This Designer Elements program provides tools and commands to transform your geometry with respect to scale, location and orientation.

This chapter covers the following topics:

- **Transformation** tools
- Transformation techniques

Transformation Tools

The tools on the **Transformation** tool palette include: move, rotate, expand or shrink and mirror objects, polar duplicate, linear duplicate and align. You can select the object you want to transform before or after you select a **Transformation** tool.

You can copy at the same time you transform objects by holding down the CTRL (Windows) or the OPTION (Macintosh) key while you specify the transformation.

When you press the SHIFT key you can select additional objects after you have selected a transformation tool (see the Message Line). As soon as you release the SHIFT key the **Transformation** tool is active again.

For each tool you can enter values in the Status Line to perform the operation either before or after the operation is complete. If you enter the values after you select the tool but before you perform the operation, your first click in the drawing area automatically registers all Status Line values. If you enter values after perform-

ing the operation and while the object is still selected, pressing ENTER (Windows) or RETURN (Macintosh) updates the object to reflect the new values.

Geometric Characteristics

There are no geometric characteristics specific to these tools. The geometric characteristics are based on the objects. For information on the geometric characteristics of an object, see its related chapter.

Move Tool

The Move tool moves selected objects, including control points to a new location. You can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key. If you select more than one object, they remain in the same position relative to each other.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when moving objects with this tool.

Tech Note:

Use this tool to create an instance of a solid rather than cutting/copying and pasting which breaks the parent/child relationship. **(Cobalt™ and Xenon™ only)**

An instance is created when you perform an operation on a solid or surface and then move it to another location. This instance object displays its own unique *Geometry* page

Using the Move Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the Move tool.
2. Select the Move tool icon in the Message Line. The Message Line reads: *Move: Select objects to move [Shift = Extend].*

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.
3. Select the object(s) to be moved.

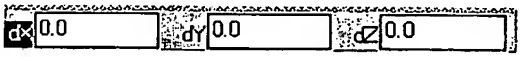
   The Message Line reads: *Move: Select beginning reference point [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*].*
4. If necessary, use SHIFT-click to select more objects, and then pick a beginning point. The Message Line now reads, *Move: Select ending reference point [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*].*
5. Click an ending reference point or drag the selected object(s) to a new location.

The reference and destination points need not be on the object(s) you are moving. The move is performed relative to the specified points.

Tech Note:

Be aware that if you use this tool on an object that has had a subtraction boolean operation, you must also move the subtracted object. Otherwise, you may alter or lose the subtraction operation.

The Status Line contains the dX, dY and dZ values of the move distance and direction.



After the move and while the object is still selected you can change a value in a data field and press ENTER (Windows) or RETURN (Macintosh) to accept the value.

Rotate Tools

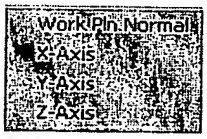
When you select the Rotate tools icon a subpalette appears in the Message Line containing three tools, Rotate by Angle (or by one point) and Rotate by Three Points.

You can use them to rotate a selected set of objects around a point in any axis, or to create a circular array.

When you rotate by one point or by three points, you can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key while you select the objects. If you select more than one object, they remain in the same position relative to each other.

Rotate 1 Pt Tool

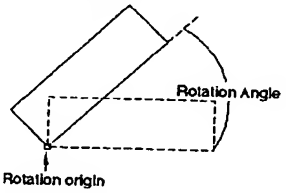
This tool rotates one or more objects around a specified point. The Message Line contains a pull-down menu for specifying the rotation axis. The options include:



Work Pln Normal	Rotates the objects about the normal vector of the work plane.
X-Axis	Rotates the objects about the X axis.
Y-Axis	Rotates the objects about the Y axis.
Z-Axis	Rotates the objects about the Z axis.

Arb. Axis

Rotates the objects about an axis you specify by picking two points on screen.



Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when rotating objects with this tool.

Using the Rotate 1 Pt Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

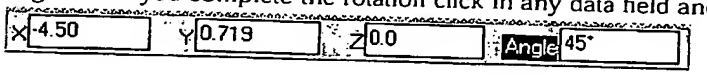
1. Select the Rotate tool.
2. Select the **Rotate 1 Pt** tool in the Message Line. The Message Line reads: *Rotate: Select items to rotate [Shift = Extend].*

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.
3. Select the objects that you want to rotate.

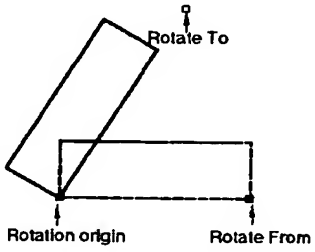
   The Message Line reads: *Rotate 1 Pt: Enter location to rotate about [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*].*
4. Use SHIFT-click to select more objects. The Message Line also includes a pull-down menu for choosing the rotation axis.
5. Select the rotation axis from the pull-down menu.
6. Specify the center of rotation.

   The object is rotated.

The Status Line contains the X, Y and Z values of the rotation point and the rotation angle. After you complete the rotation click in any data field and enter a new value.



Rotate 3 Pts Tool



The Rotate 3 Pts tool rotates an object based on three defining points, the center of rotation, a beginning reference point, and an ending reference point. This Designer Elements program determines the rotation angle from the two reference points. The two reference points need not be on the object.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when rotating objects with this tool.

Using the Rotate 3Pts Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

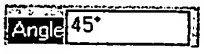
1. Select the Rotate tool.
2. Select the **Rotate 3 Pts** tool in the Message Line. The Message Line reads: *Rotate: Select items to rotate [Shift = Extend].*

   If your objects are already selected, skip to step 3. The Message Line you see reflects the previous selection.
3. Select the objects that you want to rotate.

   The Message Line reads: *Rotate: Select center of rotation [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*].* If necessary, use SHIFT-click to select more objects.
4. Click the rotation point.
5. Click the first reference point.
6. Click the second reference point.

   The object is rotated.

The Status Line contains the rotation angle.



After you complete the rotation and while the object is still selected you can enter a new value if necessary. Press ENTER (Windows) or RETURN (Macintosh) to accept the value.

Expand/Shrink and Scale Tool

The Expand/Shrink and Scale tool enlarges or reduces objects. The Message Line contains a pull-down menu with two options:

Uniform — When selected, the tool is the **Expand/Shrink** tool. This tool maintains the proportions of the stretched objects. This tool can be used for curves, surfaces and solids.

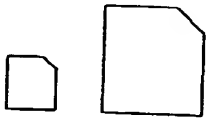
Differential — When selected this tool is the **Scale** tool and allows you to specify separate scales in the X, Y and Z axes. This tool can be used for curves, surfaces and solids.

You can copy the selection as it is scaled by holding down the CTRL (Windows) or OPTION (Macintosh) key. If you select more than one object, they remain in the same position relative to each other.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when stretching objects.

Using the Uniform Option or the Expand/Shrink Tool

This option/tool lets you expand or shrink your curves, surfaces and solids uniformly. You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability. The graphic here shows uniformly scaled curves.



1. Select the **Expand/Shrink** tool. In The Message Line reads: *Expand/Shrink: Select items to scale [Shift = Extend].*

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to scale. The option pull-down menu appears.
3. Select the *Uniform* option from the menu. When you move the pointer into the drawing area the pointer becomes the expand/shrink icon, shown here.

The Message line now reads, *Expand/Shrink: Select anchor point [Shift = Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*. If you wish to select more objects, do so.

4. Click the anchor point on your geometry. This point remains fixed in the stretch. Use the SHIFT key to select more objects to stretch for this operation.

   The Message Line now reads, *Expand/Shrink: Select beginning reference point [Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

   The beginning reference point is a point on your geometry used in conjunction with the ending reference point to set the scale of the operation. Rather than specifying reference points you can also just enter a value in the Scale data field and press ENTER (Windows) or RETURN (Macintosh) and the object will scale.

5. Click the beginning reference point.

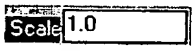
   The Message Line reads: *Expand/Shrink: Select ending reference point [Shift=Select, Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*]*.

6. Click the ending reference point. The scale factor is calculated based on the beginning reference point and this point.

   The selected object is resized. The Scale is displayed in the Status Line data field.

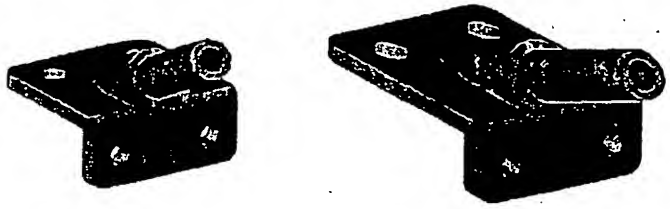
After you complete the scale and while the object is still selected you can enter a new value scale. Press ENTER (Windows) or RETURN (Macintosh) to accept the new value.

The Status Line contains the Scale factor. Entering a number between zero (0) and one (1) reduces the object. Entering a value above 1 enlarges the object.



Using the Differential Option or the Scale Tool

This option/tool lets you scale your curves, surfaces and solids differentially. You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability. The graphic here shows a differentially scaled solid.



Tech Note:

When you use this *Differential* option of the **Expand/ Shrink** tool for surfaces and solids, the stretched object displays its own *Geometry* page in Edit Objects.

1. Select the Expand/Shrink tool. In The Message Line reads: *Expand/Shrink: Select items to scale [Shift = Extend].*

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to scale.

   The option pull-down menu appears.

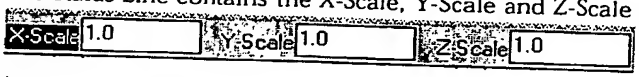
3. Select the *Differential* option from the menu. When you move the pointer into the drawing area the pointer becomes the scale icon, shown here 

   The Message Line reads: *Scale: Enter point to scale about [Shift = Select, Ctrl = Copy* (Windows) or *Option=Copy* (Macintosh)*].* If necessary, use SHIFT-click to select more objects.

4. Enter the X, Y and Z Scale values in the Status Line.

5. Click a point about which your object will be scaled.

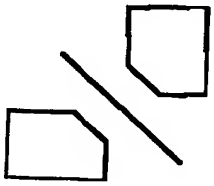
The Status Line contains the X-Scale, Y-Scale and Z-Scale data fields.



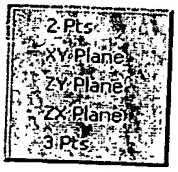
After you complete the scale, while the object is still selected, you can enter new values. Press ENTER (Windows) or RETURN (Macintosh) to accept the new values.

Mirror Tool

The Mirror tool creates a mirror image of an object or objects on the opposite side of a reference line. You can copy the selection by holding down the CTRL (Windows) or OPTION (Macintosh) key. If you select more than one object they remain in the same position relative to each other.



The Message Line contains a pull-down menu with five options for specifying the mirror reference line or plane.



2 Pts Mirrors the object around the reference created by the two specified points.

XY Plane	Mirrors the object around the XY plane.
ZY Plane	Mirrors the object around the ZY plane.
ZX Plane	Mirrors the object around the ZX plane.
3 Pts	Mirrors the object around the mirror plane specified by three points.

Tip:

When using the plane options, it may be helpful to display the Axis for reference. Choose ***Layout>Show Axis***.

Important: (Windows users) Because the CTRL and SHIFT keys are programmed to perform specific operations for this tool, the Stroke Zoom function (CTRL+SHIFT keys) is not available when mirroring objects.

Using the Mirror Tool - 2 Pts Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend].*

   If your objects are already selected, skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored.

   An option pull-down menu appears.

3. Select the *2 Pts* option in the pull-down menu. The Message Line reads: *Mirror: Select beginning of reference line [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*].*

   If necessary, use SHIFT-click to select more objects.

4. Click a location to set the beginning of the reference line around which the object mirrors. The Message Line now reads, *Mirror: Select end of reference line [Ctrl = Copy* (Windows) or *Option = Copy* (Macintosh)*].*

   If you drag you specify the beginning and endpoints and the object mirrors.

5. Click to locate the end of the reference line.

   The reference line need not be parallel to the object.

Using the Mirror Tool - XY Plane Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend].*

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored.

   An option pull-down menu appears.

3. Select the *XY Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Pt: Enter 1 point for mirror origin [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*].*

   Use SHIFT-click to select more objects.

4. Click a point in the XY plane to specify a reference point for the mirror.

Using the Mirror Tool - ZY Plane Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend].*

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored. An option pull-down menu appears.

3. Select the *ZY Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Pt: Enter 1 point for mirror origin [Shift= Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*].*

   Use SHIFT-click to select more objects.

4. Click a point in the ZY plane to specify a reference point for the mirror.

Using the Mirror Tool - ZX Plane Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend].*

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored. An option pull-down menu appears.

3. Select the *ZX Plane* option in the pull-down menu. The Message Line reads: *Mirror 1 Pt: Enter 1 point for mirror origin [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

   Use SHIFT-click to select more objects.

4. Click a point in the ZX plane to specify a reference point for the mirror.

Using the Mirror Tool - 3 Pts Option

You can select the object before or after selecting this option. The Message Line adjusts to accommodate this ability.

1. Select the Mirror tool. The Message Line reads: *Mirror: Select items to mirror [Shift = Extend]*.

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.

2. Select the object(s) to be mirrored.

   An option pull-down menu appears.

3. Select the *3 Pts* option in the pull-down menu. The Message Line reads: *Mirror 3 Pts: Enter 3 points for mirror plane [Shift=Select, Ctrl=Copy* (Windows) or *Option=Copy* (Macintosh)*]*.

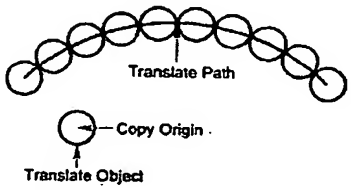
   Use SHIFT-click to select more objects.

4. Click three points to specify a reference plane for the mirror.

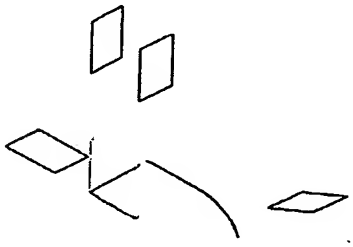
There are no Status Line entries for the Mirror tools.

Copy Along Path Tool

The Copy Along Path tool copies an object along a path by the number of times you set. The objects will be equally spaced along the curve.



You can align your copies along the X, Y and Z axis (the alignment axis). The graphic here shows a polygon created in the Top plane with a Z alignment axis.



Using the Copy Along Path Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the Move tool.
2. Select the Copy Along Path tool in the Message Line. The Message Line reads: *Copy Along Path: Select items to translate along path [Shift = Extend].*

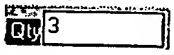
   

   If your objects are already selected skip to step 3. The Message Line you see reflects the previous selection.
3. Select the objects to copy.

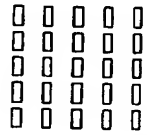
   The Message Line now reads, *Copy Along Path: Select the translation path for objects.*
4. Enter the number of objects to copy in the Qty (Quantity) field of the Status Line.
5. Click the path curve. The Message Line reads: *Copy Along Path: Enter copy origin.*
6. Click the point where the copy should begin. The Message Line reads: *Copy: Along Path: Enter two points for alignment axis.*

   You can align the copies along the X, Y or Z axis. Use the Drafting Assistant to choose the desired axis.
7. Pick two points on the screen to show your Designer Elements program where to align the object(s).

The Status Line contains the Qty (Quantity) field displaying the number of objects this Designer Elements program will create. You must set this value before copying the objects.

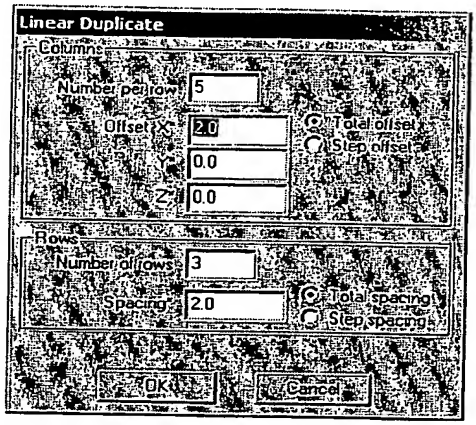


Linear Duplicate Tool



The Linear Duplicate tool creates copies of an object in a rectangular array.

To complete the array, you set the number of horizontal rows and vertical columns and the amount of space between each object in the array. This is done through the Linear Duplicate dialog box.



The dialog box contains the following options:

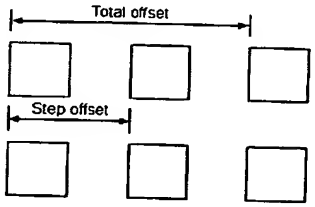
Number per row (Columns section) This field sets the total number of objects per row. Be sure to include the selected object in this total count.

If you don't know the number, you can use math operators to determine the number of copies. For example, if you want to place studs 16 inches apart on a 17 foot wall, you would enter (17*12)/16. This Designer Elements program will truncate the result to 12 copies.

Offset X*, Y*, Z* (Columns section) These fields set the offset length in the X, Y or Z direction for the duplicated objects. These values work with the Total offset and Step offset options. Enter these values yourself or drag in the drawing area to enter the values.

Total offset

(Columns section) When selected. this option specifies that the offset values in the X, Y and Z fields are the total offset values from the selected object to the last copy.



Step offset

(Columns section) When selected, this specifies that the values in the X, Y and Z fields are step offset values from the selected object to the first copy.

Number of rows

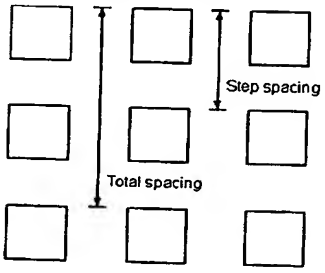
(Rows section) This field sets the total number of rows.

Spacing*

(Rows section) This field sets the perpendicular spacing for rows. This field operates with the Total spacing and Step spacing options.

Total spacing

(Rows section) When selected, this option specifies that the Spacing value is the total spacing from the selected object to the last copy. This option works with the Spacing field.



Step spacing

(Rows section) When selected, this option specifies that the Spacing value is the distance from the first

	selected object to the first copy. This option works with the Spacing field.
OK/Cancel	Click OK to complete the operation. Click Cancel to discontinue the operation.

The asterisk (*) indicates that the value automatically fills in when you drag the pointer in the drawing area to indicate the offset or spacing.

Using the Linear Duplicate (Rectangular Arrays) Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the Move tool.
2. Select the Linear Duplicate tool in the Message Line. The Message Line reads: *Linear Duplicate: Select objects for linear duplicate [Shift = Extend].*

   If your objects are already selected the Linear Duplicate dialog box appears. Skip to step 3.
3. Select the object(s) to be duplicated. The dialog box appears.

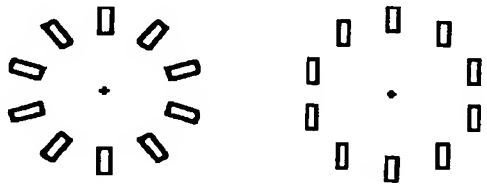
   The Message Line now reads, *Linear Duplicate: Specify parameters for linear duplicate.*
4. Enter the number of columns in the *Number per row* data field.
5. Select offset option, either Total or Step.
6. If the *Offset X** field is not selected, click in the field.
7. Move the pointer to the drawing area and drag to indicate the offset for the row of copies. Values automatically appear in the X*, Y* and Z* data fields.
8. Select the spacing option, either Total or Step.
9. If the *Spacing** field is not selected, click in the field.
10. Move the pointer to the drawing area and drag to indicate the spacing for the spacing for the rows.
11. Click OK to close the dialog box and create the duplicates.

There are no Status Line entries.

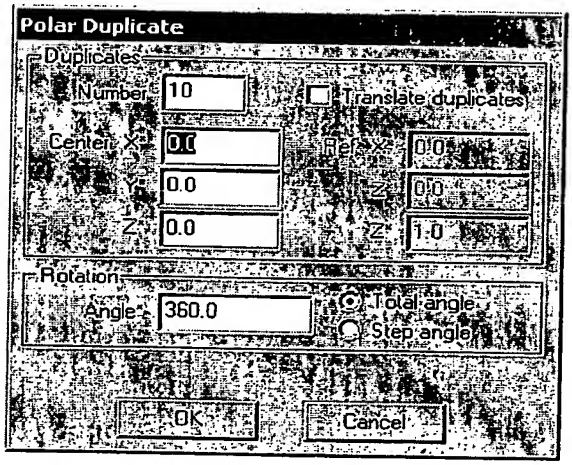
Note: This Designer Elements program does not support using this tool for features.

Polar Duplicate Tool

The Polar Duplicate tool copies a selection around a central point. The entities in the selection may be rotated as they are copied or remain upright.



The number and location of duplicates are controlled through the Polar Duplicate dialog box.



The dialog box contains the following options:

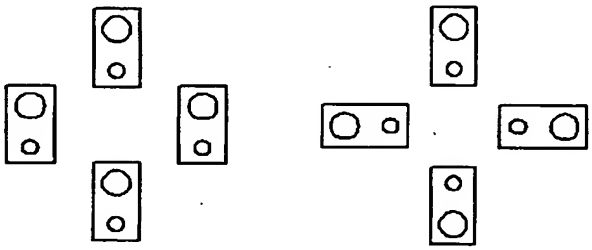
Number (Duplicates section) This field sets the total number of objects. Be sure to include the selected object in this total count.

Center X*, Y*, Z* (Duplicates section) These fields set the X, Y and Z location for the center of the circular array. You

can enter these values yourself or click the place in the drawing area for the center.

Translate duplicates

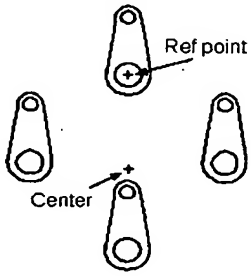
(Duplicates section) When selected, this option creates copies that are upright with respect to the original object. When this option is not selected, copies are rotated around the center you specified. The left graphic here shows an duplicated object when the option is selected. The right graphic shows a duplicated object when the option is not selected.



Ref X*, Y*, Z*

(Duplicates section) These fields represent the reference point for the Translate duplicates option.

Ref X, Y and Z determine the imaginary point duplicated around the center you specified. The selected objects are reproduced in the same position relative to each imaginary point duplicated. The imaginary Ref point and center point are indicated in the example here.



	You can specify the Ref point by entering values or clicking at the desired location in the drawing area.
Angle	(Rotation section) This field specifies the angle of rotation. This field operates with the Total angle and Step angle options.
Total angle	(Rotation section) When selected, this option specifies that the angle value is the total number of degrees between the center of the selected object and the center of the last object. (If 360° is the specified angle, the last object is the selected object.)
Step angle	(Rotation section) When selected this option specifies that the angle value is the angle between the center of two adjacent objects.
OK/Cancel	Click OK to complete the operation. Click Cancel to discontinue the operation.

The asterisk (*) indicates that the value automatically fills in when you drag the pointer in the drawing area to indicate the offset or spacing.

Using the Polar Duplicate Tool

You can select the object before or after selecting this tool. The Message Line adjusts to accommodate this ability.

1. Select the Polar Duplicate tool

2. The Message Line reads: *Polar Duplicate: Select items to polar duplicate [Shift = Extend].*

   If your objects are already selected the Polar Duplicate dialog box appears. Skip to step 3.

3. Select the object to be duplicated. The dialog box appears.

   The Message Lines reads, *Polar Duplicate: Specify parameters for polar duplicate.*

4. Enter the number of objects in the circular array.
5. If the *Center X** data field is not selected, click in the field.
6. Move the pointer to the drawing area and click the center point of the array in the drawing area. Values automatically appear in the X*, Y* and Z* data fields.

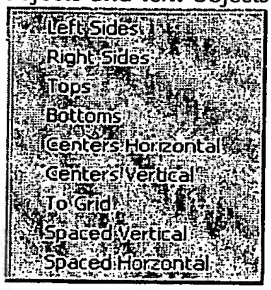
7. If you want objects to be upright with respect to the selected objects, select the *Translate duplicates* option.
8. If you specified Translate duplicates, click a location for the reference point in the drawing area.
9. Select the angle option, Total or Step.
10. Enter the angle of rotation in the *Angle* field or drag in your drawing area along the desired angle using the Draft Assistant.
11. Click OK.

There are no Status Line entries.

Note: This Designer Elements program does not support using this tool for features.

Align Objects Tool

With this Designer Elements program you have the ability to align any object created using the *Align* command. This includes aligning text objects with other text objects and text objects with geometry.



Using the Align Command

1. Choose the Selection tool.
2. Hold down the SHIFT key and select the text and geometry you want to align.
3. Choose the align tool from the Transformation tool palette.

The objects align.

For more detailed information regarding the Align Tool refer to chapeter 24, The Align Command.

Transforming Techniques

There are many ways to accomplish the same transformation task in this Designer Elements program. This section includes some of those ways.

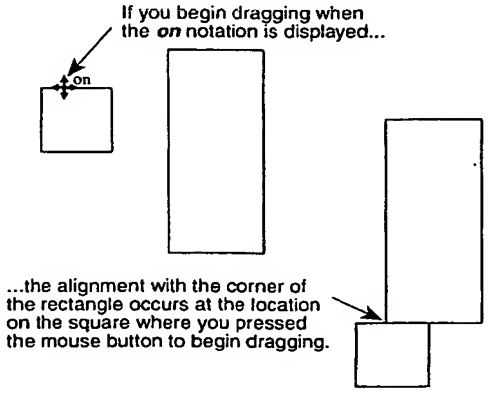
Moving Objects with Tools

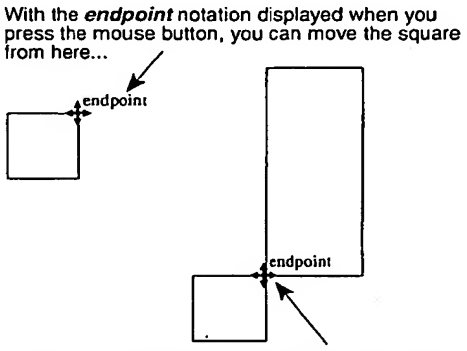
When an object is selected, move the pointer near the object until the *4-way Move* symbol appears as shown below and drag the object to a new location.

Note: If the 4-way Move symbol doesn't appear, make sure you're in the Selection tool.

Using the Drafting Assistant for Moving

When the pointer becomes the 4-way Move symbol, you can drag the object around and see the Drafting Assistant's notations relative to the object's location when you press the mouse button. This way, if you move the pointer over a control point so the Drafting Assistant activates it, you can align the object you are moving with that point.



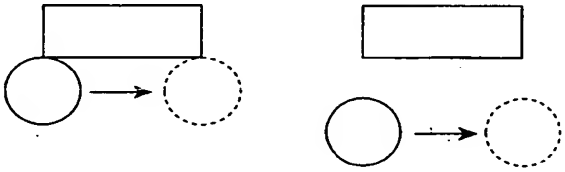


Selected Move versus Move Tool

You can move objects either with the Selection tool or with the Move tool.

Moving with the Selection Tool

The move function of the **Selection** tool allows free movement of the selected object.

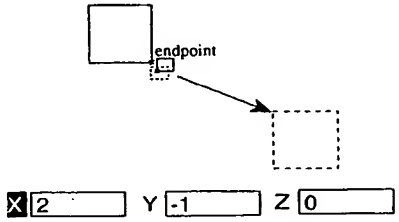


To move a circle from one location to another as shown above, use the Selection tool and the Drafting Assistant to align the 90° *quadrant* with the corner of the rectangle. The circles do not need to touch the rectangle to be aligned with it.

Moving with the Move Tool

The Move tool (in the Transformation subpalette) allows you to move the selection with reference to other geometry in the X, Y and Z direction.

Move a 1-inch square 2 inches in the X direction and 1 inch in the Y direction. See the following graphic.



1. Select the Move tool.
2. Select the square to be moved.
3. Click one corner of the square when the *endpoint* notation is displayed.
4. Enter **2** in the dX entry field on the Status Line and **-1** in the dY entry field. A positive or negative value entered in the Status Line determines the direction along the X or Y axis. A negative value moves the object to the left or down on the screen and a positive value moves the object to the right or up.
5. Press the ENTER (Windows) or RETURN (Macintosh) key. The square is moved.

Moving Objects to Another Layer

1. Choose ***Window>Edit Objects***.
2. In the Layer field, scroll down to the layer on which you want the object.
3. Click OK. The object is now located on the new layer.

Copying Objects with Tools

You can copy selections with the *Copy* command, as discussed earlier, or by holding down the CTRL (Windows) or OPTION (Macintosh) key using these tools:

- **Selection** tool
- **Single Line** tool
- **Center-Point** and **Opposite-Point Circle** tools
- **Ellipse** tools
- **Conic** tools
- **Polygon** tools

- **Project Curve** tool
- **Transformation** tools

Copying with the Selection Tool

You can hold down the CTRL (Windows) or OPTION (Macintosh) key and drag a copy of the selection to a new location.

When you use the Selection tool to copy a surface or solid using this technique you create an instance of the original object. All changes made to the original object are automatically reflected in the instance (**Cobalt™ and Xenon™ only**). If you do not want to create an instance, you can select the original object, choose ***Edit>Copy*** and then ***Edit>Paste***.

1. Choose the Selection tool.
2. Select the object(s) you want copied.
3. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
4. Drag a copy of the selection to a new location. The copy is placed on the current work layer.

Copying with the Geometry Tools

You can make copies with the following drawing tools:

- **Single Line** tool
- **Center-Point** and **Opposite-Point Circle** tools
- **Ellipse** tools
- **Conic** tools
- **Polygon** tools
- **Project Curve** tool

To create a copy with the drawing tools proceed as follows:

1. Construct the geometry you want to copy.
2. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
3. Click a new location. The click determines the location of the first point specified during the construction of the original geometry (the *center* of a Center-Point Circle, for example).

Copying with the Transformation Tools

You can hold down the CTRL (Windows) or OPTION (Macintosh) key while using a transformation tool to make a copy of the selected geometry.

1. Select one of the transformation tools.
2. Select the object you want to copy and transform.
3. Hold down the CTRL (Windows) or OPTION (Macintosh) key.
4. Perform the transformation according to the directions in the Message Line.

Copy CTRL (Windows) or OPTION (Macintosh) versus Copy Command

Normally, using the copy option of a geometry tool is faster than using the *Copy* and *Paste* commands in the Edit menu. The *Copy* and *Paste* commands are useful for copying to a different document or application.

Sizing Objects with Tools

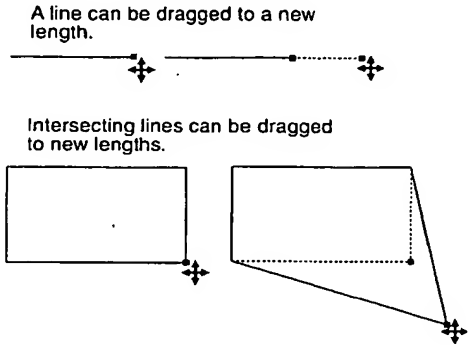
Normally you size an object with the Selection tool. In some cases it is useful to use the Move tool for sizing objects.

Sizing an Object with the Selection Tool

You can stretch objects by selecting a point and dragging it to a new location.

1. In the Edit menu, be certain that *Selectable Points* is set.
2. Click the Selection tool.
3. Drag a selection fence around the control points that represent the area you want to stretch.

4. Drag the points to a new location.

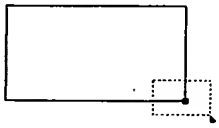


Sizing an Object with the Move Tool

You can also size an object with the Move tool. Using the Move tool allows you to specify the distance the selected point(s) should be moved by specific values along the dX, dY and dZ direction in the Status Line.

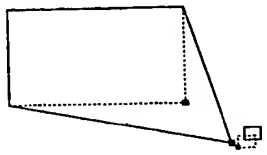
To move the corner of the single line rectangle in the next graphic with the Move tool, proceed as follows:

1. In the Edit menu, be sure that *Selectable Points* is set.
2. Click the Selection tool.
3. Drag a selection fence around the lower right corner of the rectangle.



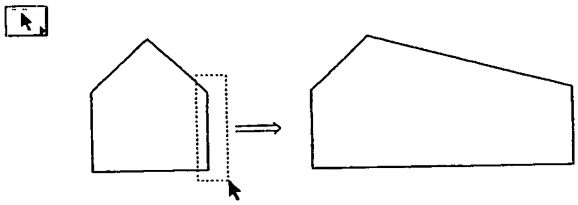
4. Select the Move tool.
5. Enter **+1** in the dX data field, a **-1** in the dY field and a **0** in the dZ field. Press the ENTER (Windows) key or the RETURN (Macintosh) key.

The corner of the rectangle is moved the distance you specified in the Status Line.

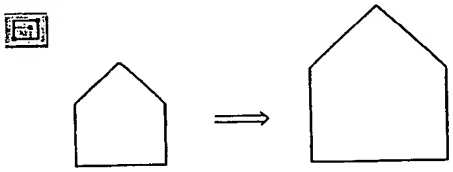


Selected Sizing versus Expand/Shrink Tool

Dragging a control point of a selected object not only changes the size of the object, but distorts the object by changing the proportion between height and width.



The **Expand/Shrink** tool on the Transformation subpalette enables you to resize geometry while maintaining its proportions.



Use the Scale tool to enlarge or shrink a shape proportionally

In addition, you can specify proportions by clicking points on other geometry. For example, you can resize an object to fit within another object by clicking the boundary into which the resized object must fit.